1/10/2012

363-S USER'S MANUAL

Revision 1.6

TABLE OF CONTENTS

Introduction / using this manual 5
Safety 6
READ BEFORE OPERATING THIS MACHINE 6
Observe and Obey All Warnings Listed Below 7
Machine Information 9
General Machine Specifications 9
Machine Requirements 10
Operating Conditions 10
Spindle Specifications 11
Software introduction 13
Touchscreen Alignment Procedure 14
On-Screen Keyboard 15
Menu Buttons 16
Panels 21
Dashboard 21
Control Panel 22
View Panel 29
Input Panel 37
Hardware Introduction 44
Coordinate Axes 44
Pallet and Spindle Mounting 45
Work Piece Mounting System 46
Spindle Introduction 48
40 000 (NR-5041E ATC) & 50 000 (NR40-5100 ATC) RPM Electric Motor Driven Spindles with Automatic Tool Changing Capability (pictured in Figure 5.4) 48
50 000 (IBAG - HF45A60CP) RPM Electric Motor Driven Spindles with Automatic Tool Changing Capability 48
80 000 RPM Electric Motor Driven Spindles (NRA-5080) 48
160 000 RPM Air Turbine-Air Bearing Spindle (ABS-1600) 48
Front of the Machine 51
Handheld Pendant 52
Machine Regulators 53
Back of the Machine 55
Installation / transportation 57
Installation 57
Transportation 58

Pallet Jack or Fork Lift 58
Operation 59
Power-On Procedure 59
Power-Down Procedure 59
Software Startup and Homing the Motors 59
Spindle Operation 60
Before Enabling the Spindle 60
Starting the Spindle 60
Changing the Spindle Speed 61
Disabling the Spindle 61
Tool Changing – Manual Collet – 80 000 RPM Electric/Air Bearing 62
Tool Installation 62
Tool Removal 63
Tool Changing–Manual Collet–160 000 RPM Air Turbine/Air Bearing 64
Tool Installation (Figure 7.6) 64
Tool Removal (Figure 7.6) 65
Tool Changing – Automatic Collet 66
Tool Installation 66
Tool Removal 67
Tool Changing - Automatic Tool Changer 68
Manual Operation Mode 68
Operation in an NC program 71
Tool Changer Alignment 71
Tool Collar Setting using Bohan Tool Setter 73
Tool Collar Setting using Two-Piece Setting System 74
Tool Sensing – G120 Cycle 75
Syntax and Parameters 75
Length Measurement Examples: 75
Calculation of X and Z parameters: 75
Auto-Sensing Multiple Tools for a Job using the Microlution mMT Control Software 75
Manual Operation Mode 76
Implementation in an NC Program 76
Z-Axis Fixture Probing – G123 Cycle 77
Syntax and Parameters 77
Fixture Probing Example 77
Workpiece Probing 78
X Y Z single surface measurement (G65 P9811) 79

Pallet ID function (G65 P9809) 81
Error code index table 84
Example 85
Subroutines and Macro Program Calls 87
Setup and Organization 87
Syntax and Parameters 87
Subprogram Parameter Variables 88
Subprogram Example 89
Data Setting 90
Data Setting Group Descriptions 90
Data Setting Examples 90
Conditional Statements 92
If ({condition}) {action} 92
While ({condition}) {action} 93
GOTO{data} 93
Pallet Changing 95
Removing the pallet 96
Installing the pallet 97
Tool Setting - Camera 98
Using the Handheld Pendant 104
Executing a Program 105
Manually Running a Program by Using the Terminal Panel 105
Automatically Running a Program by Using the Combine Tab and the Program Tab 107
Part Offsets 108
Save Current Location to a File 108
Recall Saved Location from a File 109
Set Known Location as Program Origin 109
Cleaning/Maintenance 110
Cleaning 111
Maintenance 113
Supported G and M Codes 116
Appendix a – Legacy / Custom Order 129
Spindle Changing (Kinematic Mount Type) 129
Removing the Spindle 129
Installing the Spindle 130
Workpiece sensing 130
Manual Operation Mode 131

Operation in an NC Program 131
Spindle Control Unit 133
Part Pattern Tab 136
Creating a Part Pattern 138
Legacy Cleaning / Maintenance 141
Laser Inspection (LT-9010 Confocal Laser Option Required) 142
Scanning Using a Predefined Scanning Pattern 142
Scanning Using a User Defined Scanning Pattern 143
User I/O (option required) 145
Digital Inputs 146
Digital Outputs 146
Analog Inputs 146
Filtered PWM Analog Outputs 147
User Flags 147

INTRODUCTION / USING THIS MANUAL

Thank you for purchasing the Microlution, Inc. 363-S micro-milling machine tool. This instructional manual will introduce the 363-S, its specifications, components, software, and other aspects of the machine.

Before operating the 363-S, this manual should be read thoroughly and in its entirety. After the manual has been completely read, keep it as a reference for later use.

Please read this manual from beginning to end starting with the Safety section. Following the Safety section is the Machine Information section with machine specifications, requirements, operating conditions, and other useful information. Following the Machine Information section is the Software Introduction section. The Software Introduction is where the machine software is introduced and all the buttons, functions, and windows are explained.

Note: A step – by – step set of instructions for operating the machine and navigating the software is located in the Operation section of this manual.

Following the Software Introduction is the Hardware Introduction. The Hardware Introduction introduces the exterior components of the machine and their functionality. Next, the Installation / Transportation section describes proper installation and transportation steps, precautions, and techniques. Following the Installation / Transportation section is the Operations section. The Operations section provides step – by – step instructions for operating the machine and using the software. Finally, the Cleaning / Maintenance section describes the correct procedure for cleaning and maintaining the machine.

Note: The black highlighted titles differentiate sections. The grey highlighted titles breakup the sections into sub-sections, and are therefore referred to as sub-sections. Underlined headings within sub-sections are referred to as segments.

SAFETY

READ BEFORE OPERATING THIS MACHINE

- Only authorized personnel should work on this machine. Untrained personnel present a hazard to themselves and the machine, and improper operation will void the warranty.

- Check for damaged parts and tools before operating the machine. Any part or tool that is damaged should be properly repaired or replaced by authorized personnel. Do not operate the machine if any component does not appear to be functioning correctly. Contact your shop supervisor.

- Use appropriate eye and ear protection while operating the machine. ANSI-approved impact safety goggles and OSHA-approved ear protection are recommended to reduce the risks of sight damage and hearing loss.

- Do not operate the machine unless the protective lid is closed. Rotating cutting tools can cause severe injury. When a program is running, the X,Y, and Z stages as well as the spindle can move rapidly at any time in any direction.

- The Emergency Stop button (also known as an Emergency Power Off button) is the large, circular red switch located on the handheld pendant (see Hardware Introduction section). Pressing the Emergency Stop button will shut off power to the X, Y, and Z stage motors as well as the spindle. Use the Emergency Stop button only in emergencies to avoid crashing the machine.

- All machine panels should be closed and secured at all times except during installation and servicing. During installation and servicing, only qualified personnel should have access to the machine. When power is supplied to the machine some components operate at high temperatures. Therefore, extreme caution is required. Once the machine is installed, the control cabinet must be secured and only opened by qualified service personnel.

- DO NOT modify or alter this equipment in any way. If modifications are necessary, all such requests must be handled by Microlution, Inc. Any modification or alteration of any Microlution milling machine could lead to personal injury and/or mechanical damage and will void your warranty.

- It is the machine supervisor's responsibility to make certain that all who operate or service the machine are thoroughly acquainted with the installation, operation, and safety instructions BEFORE operating or servicing the machine.

- **This machine is automatically controlled and may start at any time.**
- **This machine can cause bodily injury.**
- **Do not operate with the protective lid open.**

Observe and Obey All Warnings Listed Below

- **Do not operate without proper training.**

- **Personal Protective Equipment:**

    - **Always wear safety goggles;**
    - **Abrasion-resistant gloves are recommended when handling sharp tools, workpieces and workpiece pallets. Caution must be taken as gloves may become entangled in rotating or moving parts within the machine.**

- **The electrical power must meet the specifications in this manual. Attempting to run the machine from any other source can cause severe damage and will void the warranty.**

- **DO NOT power up machine until after the installation is complete.**

- **NEVER service the machine with the power connected.**

- **Improperly fastened parts machined at high speeds/feeds may be ejected and puncture the protective lid. Machining oversized or marginally fastened parts is not safe.**

- **The window in the protective lid must be replaced if damaged or severely scratched - Replace damaged windows immediately.**

- **Do not process toxic or flammable material. Deadly fumes can be present. Consult material manufacturer for safe handling of material by-products before processing.**

- **The machining process may generate fine, dust-like swarf. Depending on the workpiece material, appropriate care (for example vacuum swarf removal) must be taken to properly handle the swarf. Consult material manufacturer for additional information.**

- **The machine must be periodically cleaned to remove residual swarf. The user is responsible for establishing a cleaning process and schedule, as required by material removal rates, workpiece material, and other factors.**

WARNING
Hazardous voltage inside.
Remove power before servicing.

WARNING
Moving parts can crush and cut.
Do not operate with guard removed.
Follow lockout procedure before servicing.

CAUTION
Laser radiation.
DO NOT STARE INTO BEAM.
Class 2 laser product.

MACHINE INFORMATION

Warning: Read this section before operating machine! Failure to do so can result in machine malfunction and/or operator injury.

General Machine Specifications

Table 3.1: General Machine Specifications

Height	135 cm [53"]
Width	71 cm [28"]
Depth	64 cm [25"]
Footprint	0.45 m^2 [4.86 ft^2]
Weight	340 kg [750 lbs] without accessories
Maximum Feed Rate	8000 mm/min [314.9 IPM]
Machine Travel	63 mm x 63 mm x 63 mm [2.48" x 2.48" x 2.48"]
Resolution	0.05 microns [0.0000008"]
Positioning Accuracy	2.0 microns [0.00008"]

Machine Requirements

Table 3.2: Machine Requirements

Air Requirements	Minimum of 170 liters/min @ 0.7 MPa [6 CFM @ 100 PSI] supplied to the air regulator at the rear of the machine. Air supply should be clean and dry. **NOTE**: Check that the air regulator at the rear of the machine is set to 90 psi. **NOTE**: Air consumption will vary based on spindle selection and accessories.
Electrical Requirements	The machine plugs into a standard 110 VAC outlet. **NOTE**: The machine requires 20 Amps. Ensure the circuit is rated for 20 Amps.

Operating Conditions

The Microlution 363-S is a high performance precision micro-manufacturing machine tool, and as such must be operated under specific environmental conditions. Only operate machine **indoors,** in a **safe location,** free from explosive vapors and/or flammable liquids.

Table 3.3: Operating Conditions

Preferred Operating Temperature Range	20.0 °C to 22.2 °C [68 °F to 72 °F]
Operating Temperature Range	15.6 °C to 26.7 °C [60 °F to 80 °F] **NOTE**: Operating machine at a temperature not in the preferred operating temperature range may affect the accuracy of the machine.
Storage Temperature Range	0 °C to 45 °C [32 °F to 113 °F]
Ambient Humidity	The relative humidity should be maintained within a range of 50% to 65%, non condensing.
Altitude	0 m to 1829 m [0 ft – 6000 ft]

Spindle Specifications

Microlution, Inc. offers different spindle options for the 363-S. The specifications for the most common spindles offered are listed in Tables 3.4 through 3.8. Please see the "Hardware Introduction" section for more information on the different applications of each spindle.

Table 3.4: 50 000 RPM Electric Motor Driven High-Precision Bearing Spindle (NR40-5100 ATC) Specifications

Maximum Speed	50 000 RPM
Air for Collet Opening	0.55 MPa – 0.61 MPa [80 PSI - 87 PSI]
Spindle Accuracy	< 2 microns [0.00008"]
Weight	0.74kg [1 .63 lbs]
Power Consumption Rating	18 A (30)
Maximum Output Power	350 W

Table 3.5: 40 000 RPM Electric Motor Driven High-Precision Bearing Spindle (NR-5041E ATC) Specifications

Maximum Speed	40 000 RPM
Air for Collet Opening	0.50 MPa - 0.56 MPa [73 PSI - 87 PSI]
Spindle Accuracy	< 2 microns [0.00008"]
Standard Size Collet Chuck	3.175 mm [1/8"] diameter
Weight	1.34 kg [2.95 lbs]
Power Consumption Rating	18 A (30)
Maximum Power	350 W

Table 3.6: 50 000 RPM Electric Motor Driven High-Precision Bearing Spindle (IBAG-HF45A60CP) Specifications

Maximum Speed	50 000 RPM (60 000 RPM with liquid chilled spindle block)
Air for Collet Opening	0.55 MPa - 0.7 MPa [80 PSI - 101 PSI]
Pressure Air Seal	0.2 MPa
Spindle Accuracy	< 2 microns [0.00008"]
Standard Size Collet Chuck	3.175 mm [1/8"] diameter
Weight	2.3 kg [5.1 lbs]
Continuous Power	500 W
Maximum Power	975 W

Table 3.7: 100 000 RPM Ceramic Hybrid Ball Bearing Spindle (Jaeger: Z33-D11100.02S2Y) Specifications

Maximum Speed	100 000 RPM
Air for Collet Opening	0.55 MPa - 0.7 MPa [80 PSI - 101 PSI]
Standard Size Collet Chuck	3.175 mm [1/8"] diameter
Weight	1.1 kg [2.4 lbs]
Continuous Power	360 W
Maximum Power	410 W

Table 3.8: 80 000 RPM Electric Motor Driven Air Bearing Spindle (NRA-5080) Specifications

Maximum Speed	80 000 RPM
Air Consumption for Air Bearing	100 liters/min @ 0.49 MPa [3.53 CFM @ 71 PSI]
Spindle Accuracy	< 1 micron [0.00004”]
Standard Size Collet Chuck	3.175 mm [1/8"] diameter
Weight	0.96 kg [2.1 lbs]
Maximum Power	300 W

Table 3.9: 160 000 RPM Air Turbine-Air Bearing Spindle (ABS-1600) Specifications

Maximum Speed	160 000 RPM
Proper Air Pressure for Air Turbine	0.29 MPa – 0.49 MPa [42 psi-71 psi]
Air Consumption for Air Turbine	100 liters/min @ 0.49 MPa [3.53 CFM @ 71 PSI]
Proper Air Pressure for Air Bearing	0.49MPa [71 PSI]
Air Consumption for Air Bearing	40 liters/min @ 0.49MPa [1.41 CFM @ 71 PSI]
Spindle Accuracy	< 1 micron [0.00004”]
Radial Bearing Load Capacity	< 6 N @ 0.49 MPa [1.35 lbf @ 71 PSI]
Thrust Bearing Load Capacity	< 6 N @ 0.49 MPa [1.35 lbf @ 71 PSI]
Standard Size Collet Chuck	3.175 mm [1/8"] diameter
Weight	0.91 kg [2 lbs]
Maximum Power	14 W

SOFTWARE INTRODUCTION

Upon opening the Microlution 363-S CNC Control Software, the main interface window is visible as indicated in Figure 4.1. This window is divided into four main panels and a menu bar across the top. Each button on the menu bar will open a separate window containing additional functionality. The panels in the main interface window are used for direct control of and feedback from the Microlution 363-S CNC machine tool. Located within each panel are several tabs as shown in Figure 4.1. A tab can be toggled by pressing the corresponding button at the top of its panel. The functionality of each tab as well as each panel is described in this section. Also described in this section is the functionality of each control button located within the various tabs of the Microlution 363-S CNC Control Software. A step – by – step set of instructions on how to operate the 363-S machine tool through the CNC Control Software is detailed in the Operation section of this manual.

The remainder of this section is organized as follows. First, instructions for how to use the on-screen keyboard are given. Next, the menu buttons and their corresponding windows are detailed. Next, the four panels are presented, and the tabs within each of these panels are described in detail. Also included in this discussion is a description of the functionality of each of the buttons within the tabs.

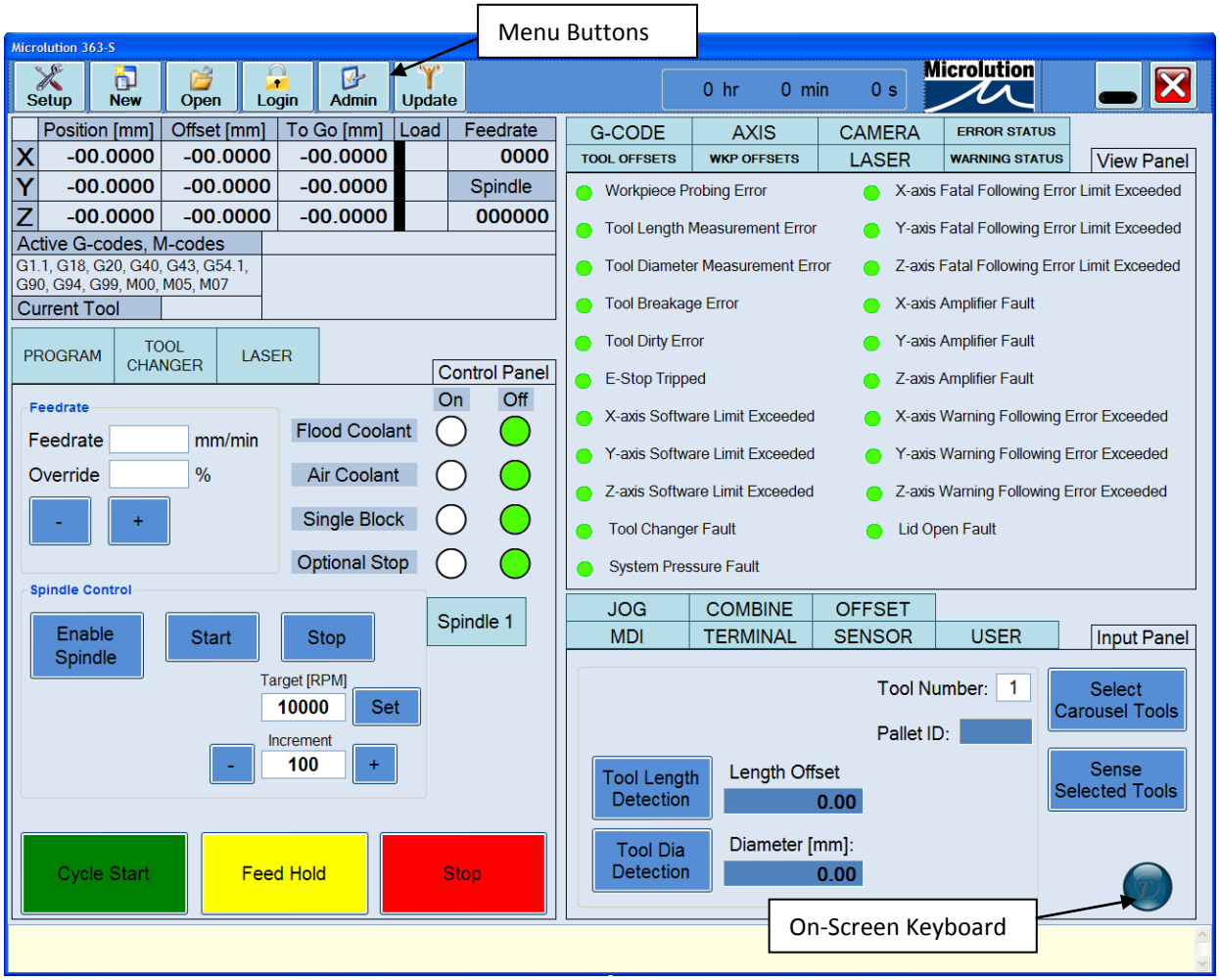
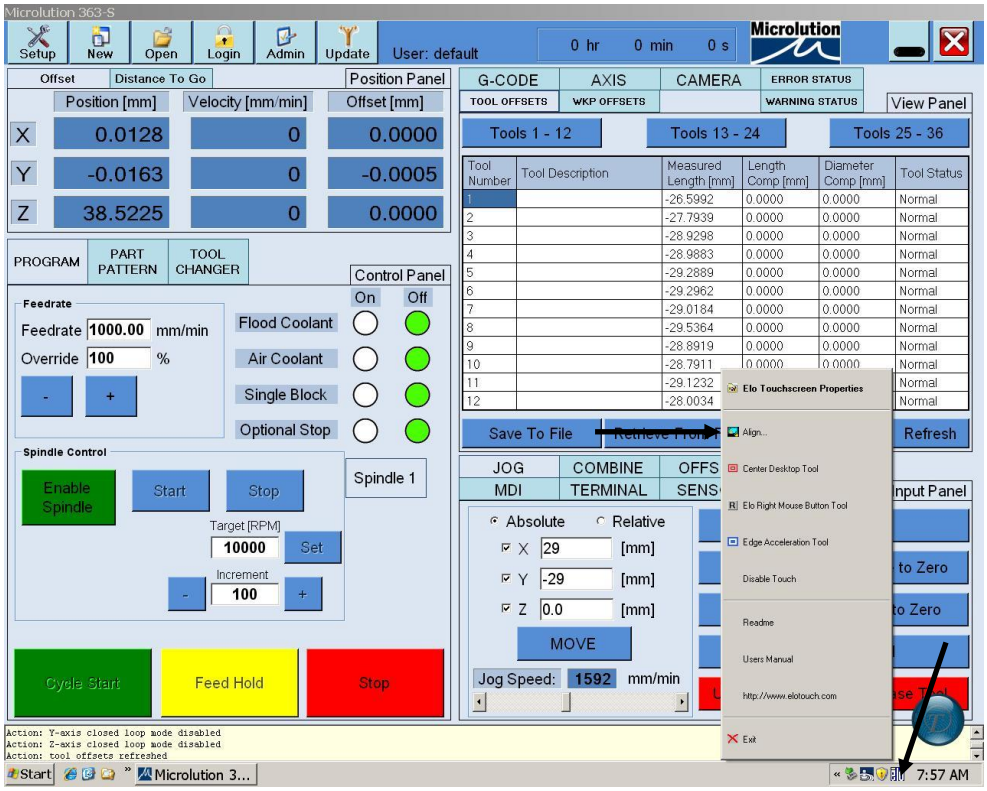


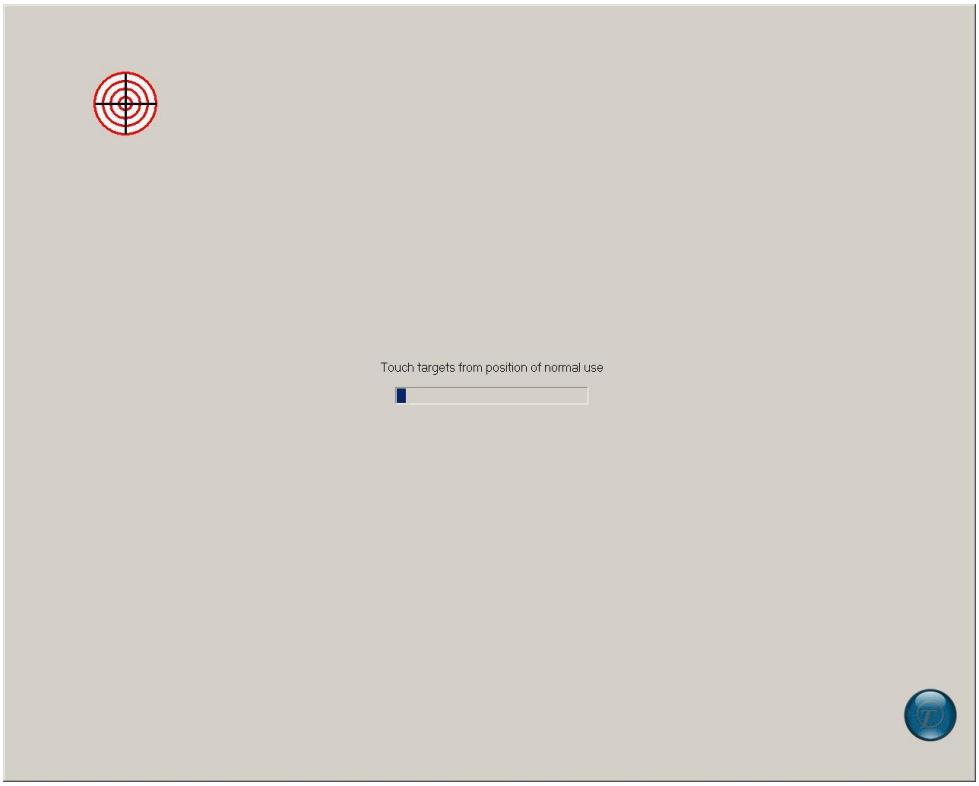
Figure 4.1: Program Start-up Screen with CNC Control Panel, Dashboard, View Panel, Input Panel and Menu Buttons in the top row. Also, the on-screen keyboard button is shown in the lower-right of the screen.

Touchscreen Alignment Procedure

1. Click/touch the *Elo* icon in the system tray on the bottom-right side of the screen. In the pop-up menu, select *Align…*



2. Touch the center of the targets that pop up on the screen.



3. Repeat steps 1 and 2 until you are satisfied with the alignment.

On-Screen Keyboard

A third-party software package is used as an on-screen keyboard to input characters and commands into the machine tool's computer. The keyboard is initiated upon startup of the machine. Figure 4.2 shows the layout of the on-screen keyboard. Table 4.1 gives a description of the menu buttons on the keyboard. For more information about the touch-screen keyboard, you can visit the software's official website at http://www.chessware.ch.

Table 4.1: On-screen keyboard menu button descriptions

Button	Image	Function
Menu		Pulls up the main menu for the keyboard. It is usually a good idea to change the mode to *Floating* after startup so that the keyboard can be dragged anywhere on the screen.
Right-Click		After pressing this button, the next time the screen is touched a right-click will be simulated. After the right-click occurs this button is no longer enabled, and any subsequent touch of the screen will result in a normal left-click.
Switch Language		Allows the user to switch the keyboard keys to a different character set. Currently, English is the only language available.
Hide		Hides the keyboard from view. The keyboard can be unhidden by pressing the button in the lower right of the screen that shows up while the keyboard is hidden.
Power		Turns off the keyboard. The easiest way to reopen the keyboard after turning it off is through the *Start* menu.

Figure 4.2: On-screen keyboard

Menu Buttons

Additional panel tabs are located at the top of the main Start-up screen as labeled in Figure 4.1. Clicking on the icon brings up the panel.

Setup

The Setup menu button is located in the upper left corner of the main interface window at startup. The Setup Window (Figure 4.3) is used for initial motor homing, motion setup, and may contain customized setup software (see Appendix A for customized setup instructions). Upon machine power-up, the motors must be phased. See the *Software Startup and Homing the Motors* sub-section under the *Operation* section of this manual for instructions on this procedure.

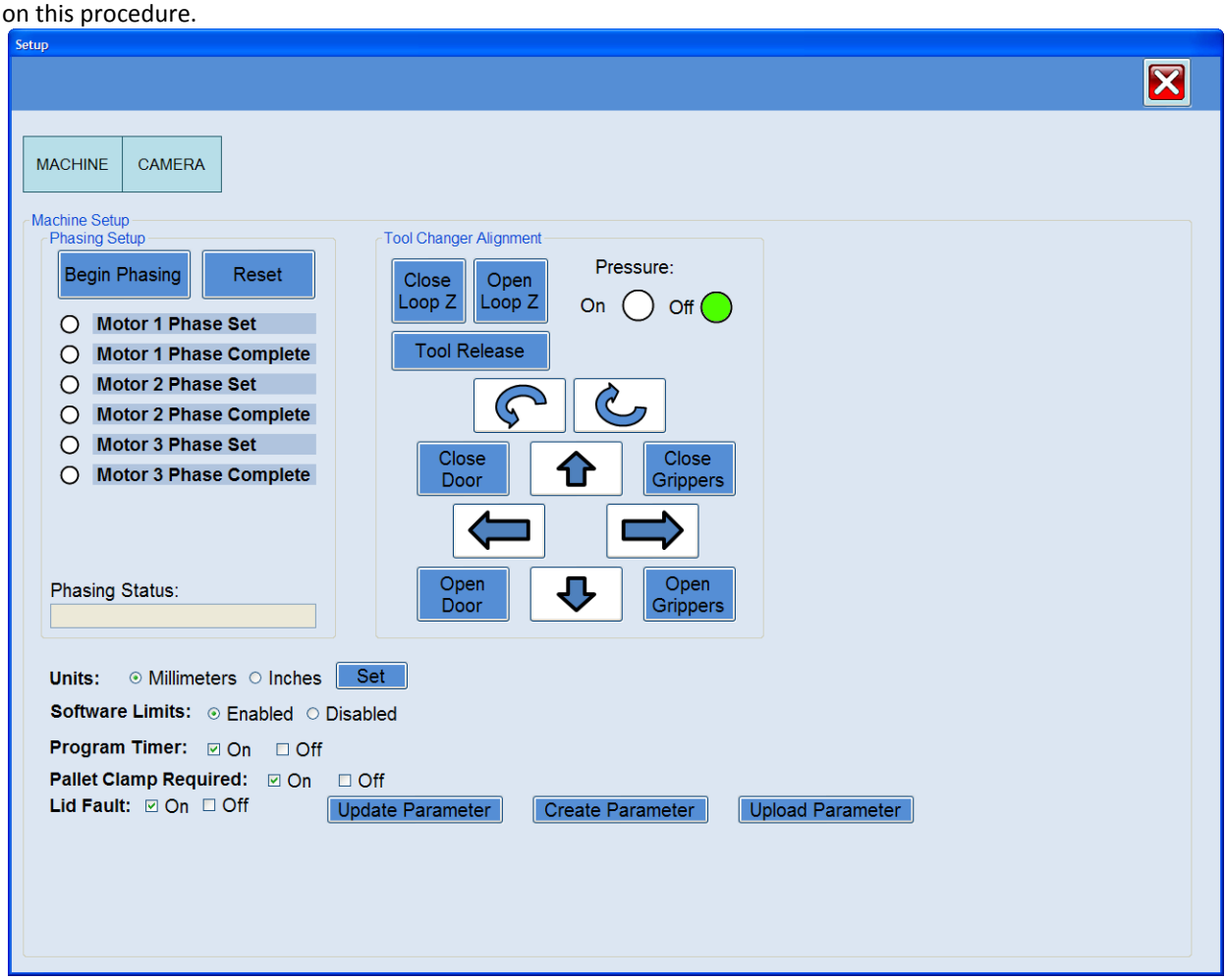


Figure 4.3: Setup Window

New Program

Clicking the *New* menu button will open a simple text editor, shown in Figure 4.4, to write short programs. The editor also has the ability to download the program directly to the 363-S memory by pressing the *Download* button. The editor is shown in Figure 4.4.

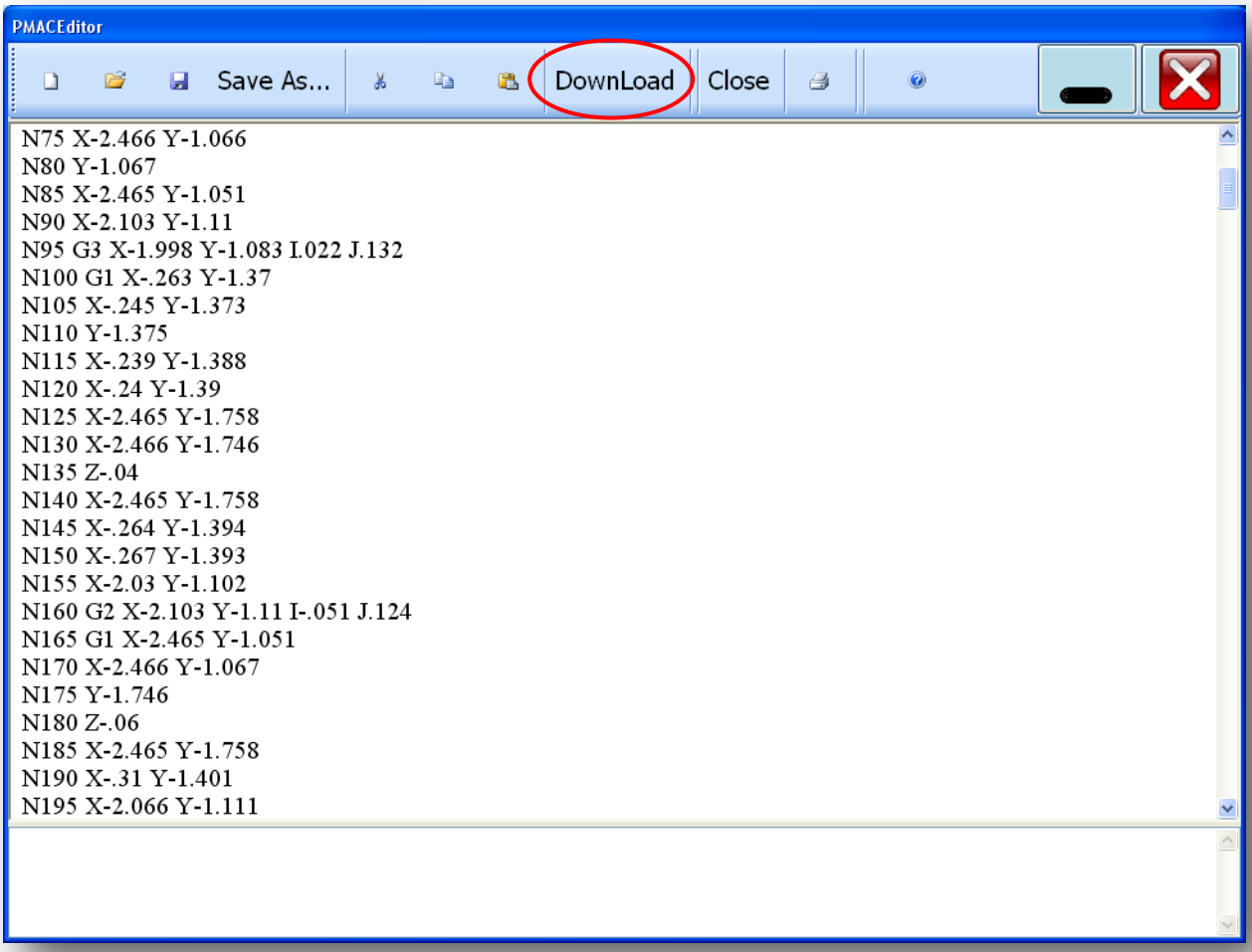


Figure 4.4: PMAC Editor

Open Program

The *Open* button will open a dialog box allowing the user to browse for a preexisting file. After selecting the desired file clicking the *Open* button in the dialog box will load the file into the editor.

Login

The *Login* menu button will open a window where a user login name and password can be entered (shown in Figure 4.5). The default user is loaded upon startup and this button must then be used in order to change users.

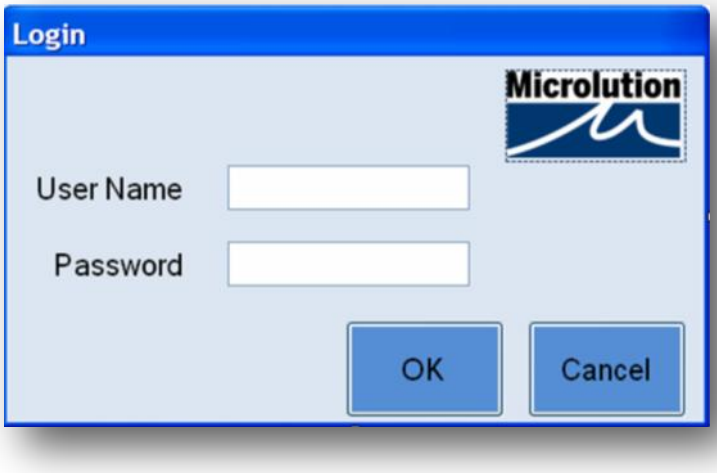


Figure 4.5: User login window

Administration

The *Admin* menu button will open a new window for user account setup (shown in Figure 4.6). By double clicking on a user account in the user account list box, the controls that user has access to can be modified. On the right side of the window is a list of all available controls. These controls can be toggled on/off for each user. The controls that are deactivated can be either shown as disabled or hidden depending on the radiobutton that is toggled. Also, the user login name and password can be changed in their corresponding text boxes. After any modifications have been made to a user account, the *Save* button must be pressed to implement and save those changes. The controls for additional add-on features that have not been purchased will be disabled. The Default and Admin users are permenant and cannot be deleted.

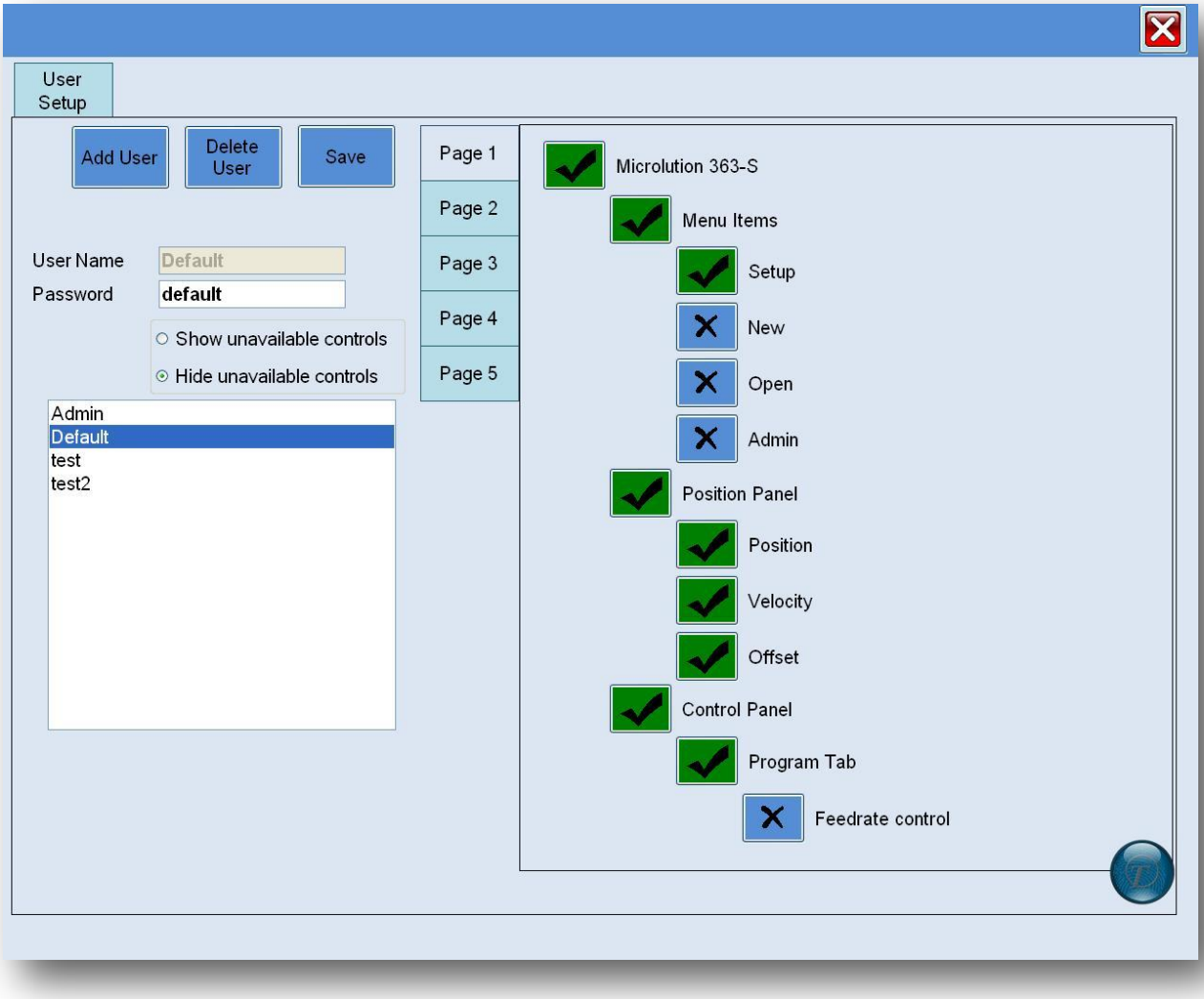


Figure 4.6: Administration window for user account setup

Update

The Update menu button will open a new window, shown in Figure 4.7 where a 25 digit access code can be entered to enable any 363-S mMT add-on features. Popular add-on features include a laser distance sensor, a tool changer, a camera vision system, and several different spindle options to name a few. Upon purchasing one of these add-on features, an access code will be supplied by Microlution, which can be entered in this window to unlock the corresponding software controls. Confirmation that the upgrade was successful will be given upon entering a correct access code. In order to use the newly available software controls, they must be activated in the user account setup window for each user.

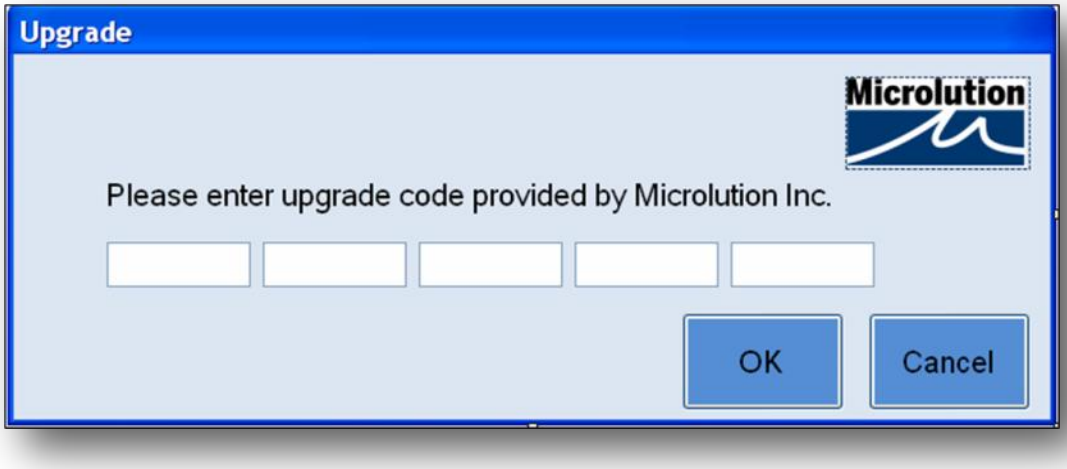


Figure 4.7: Upgrade window

Panels

There are four panels in the main interface window of the Microlution 363-S CNC Control Software as shown in Figure 4.1. The panels are the Dashboard, Control Panel, View Panel, and Input Panel. Each panel, except the Dashboard, has several tabs where similarly grouped controls are contained. Each tab can be accessed by pressing its corresponding botton at the top of its panel. The remainder of this section describes in detail the tabs in each panel and the controls contained in each tab.

Dashboard

Start-up opens the Dashboard in the upper left corner as shown in Figure 4.8. This panel shows the current state of the machine. The top sections shows the current location of the axes in program coordinates, the current offset of the axes from its initial homing coordinates, the distance-to-go to the commanded position each axis, the load on each motor, the vector feedrate, and the spindle speed. The bottom section shows the active G-codes and M-codes and the tool that's currently in the spindle.

	Position [mm]	Offset [mm]	To Go [mm]	Load	Feedrate
X	-00.0000	-00.0000	-00.0000		0000
Y	-00.0000	-00.0000	-00.0000		Spindle
Z	-00.0000	-00.0000	-00.0000		000000
Active G-codes, M-codes					
G1.1, G18, G20, G40, G43, G54.1, G90, G94, G99, M00, M05, M07					
Current Tool					

Figure 4.8: Dashboard

Control Panel

The panel in the bottom right corner of the Main Interface Window is the Control Panel as seen in Figure 4.9. The Control Panel is the primary interface through which the user runs CNC programs and operates the laser and tool changer if available. The four tabs in this panel are for program running, part patterning, laser operation, and manual tool changer setup and operation.

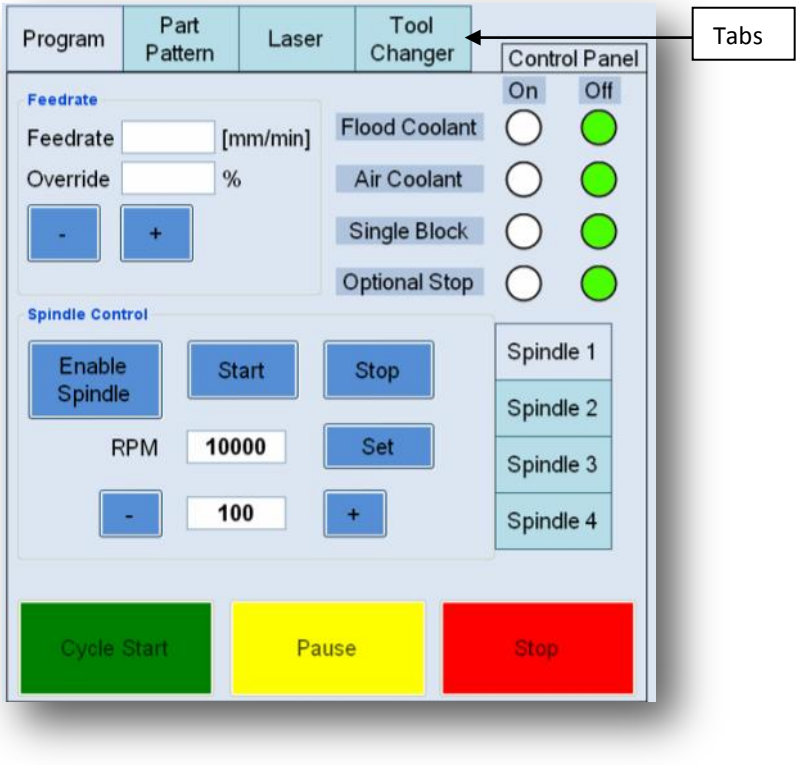


Figure 4.9: Control Panel

Program Tab

The Program Tab, Figure 4.10, is used to run a single pre-existing G-code program on the 363-S. It is also used to control the spindle operation, feedrate, coolant, and program run modes. Table 4.2 lists the different buttons located in the Program Tab and the functionality of each button. Refer to the Operation section of this manual for a step – by – step set of instructions for using the Program Tab.

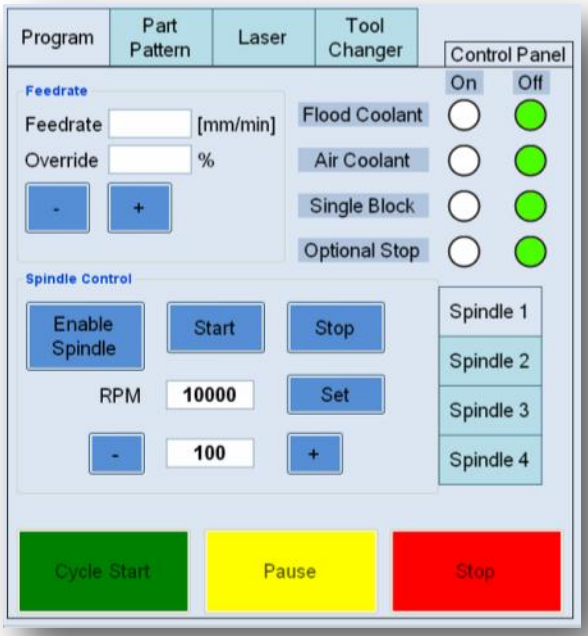


Figure 4.10: Program Tab

Table 4.2: Program Tab buttons and their functions

Button	**Function**
Cycle Start	Executes the program from the first line. If the program has been paused it will resume from the paused line.
Pause	Pauses the program following the execution of the line during which the Pause button was pressed. Pressing *Cycle Start* will resume the program from the paused location.
Stop	Halts execution of the G-code and motion of the machine even if it is in the middle of a move. Pressing *Cycle Start* after pressing the *Stop* button will resume the current move and then execute the next line.
Flood Coolant	Starts or stops flood coolant flow. Air coolant must be turned off to turn on flood coolant.
Air Coolant	Starts or stops air coolant flow. Flood coolant must be turned off to turn on air coolant.
Single Block	This changes the function of *Cycle Start*. If *Single Block* is 'on' each time Cycle Start is pressed the machine will only execute the next program line. If *Single Block* is 'off' the program will continue executing until stopped or completed.

Optional Stop	When optional stop is 'on', program execution will pause when an M01 command is given in the CNC program. If optional stop is turned off, all M01 commands will be ignored during program execution.
Feedrate	The text box will display the feedrate from the G-code. The Override box is used to scale the G-code feedrate commands.
Override	Override will increase or decrease the displayed feedrate by the percentage displayed in the Override text box. This can be useful to slow down a program and monitor all operations to ensure they are completed as anticipated or speed up the program to determine optimum feedrates. The maximum override is 200% of the displayed feedrate. **Note**: Changing the Override will not change the feedrate displayed in the Feedrate Text Box. However, the displayed feedrate is scaled by the percent listed in the Override Text Box. The true feedrate is the scaled override value of the displayed feedrate in the Feedrate Text Box.
Enable Spindle	Turns the air supply to the air bearings in the spindle on. When the spindle is enabled the speed can be set and the spindle started and stopped, the tool release is also disabled so that a tool cannot be released while in motion.
RPM	The RPM Text Box displays the desired spindle speed. The spindle must be enabled to modify the RPM text box. Click the *Set* button to set the spindle speed to the displayed value. The spindle must be enabled and on to adjust spindle speed. The *+/-* buttons will increase or decrease the spindle speed by the displayed increment.

Laser Inspection Tab (LT-9000 Confocal Laser Option Required)

The Laser Inspection Tab (see Figure 4.12) allows the user to scan a part using a built-in scan path or by using a user-defined scan path. Table 4.4 lists the applicable buttons and their functions for the Laser Inspection Tab. Refer to the Operation section of this manual for a step – by – step set of instructions for using the Laser Inspection Tab.

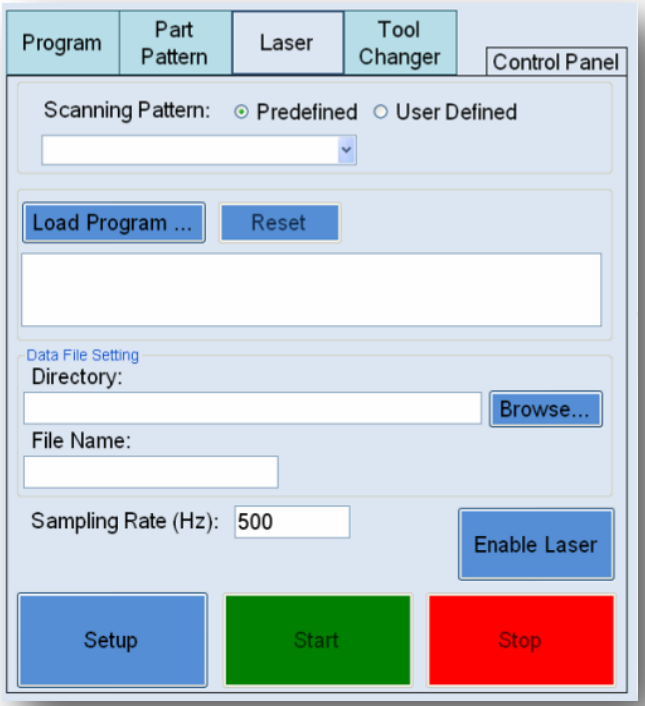


Figure 4.12: Laser Inspection Tab

Table 4.4: Laser Inspection Tab buttons and their functions

Button	Function
Predefined Scanning Pattern	Select pattern to use.
Rectangular Area (X Scan) (Pulldown Menu)	Sets the scan to proceed from the start X and Y position to the end X position. After reaching the end X position, the Y position is incremented towards the Y end position by the step amount and scans from the end X value to the start X value. This process repeats until the end value of X and Y is reached.
Rectangular Area (Y Scan) (Pulldown Menu)	Sets the scan to proceed from the start X and Y position to the end Y position. After reaching the end Y position the X position is incremented towards the X end position by the step amount and scans from the end Y value to the start Y value. This process repeats until the end value of X and Y is reached.
Circular Area (R Scan) (Pulldown Menu)	The scan will move radially starting at the start radius and start angle until the end radius is reached. The angle will be incremented by the step amount and the scan will proceed along the new radius until reaching the start radius. This process repeats until the end value of radius and angle is reached.
Circular Area (Arc Scan) (Pulldown Menu)	The scan will move along an arc starting with a radius equal to the start radius and angle equal to the start angle. The angle will increment until the end angle is reached. The radius will then be incremented by step amount. The scan will then decrement the angle until the start angle is reached. This process repeats until the end value of radius and angle is

	reached.
User Defined Scanning Program	Allows the user to load a G-code program to control the scan, useful for more complex scans or sequentially completing multiple types of scans.
Sample Spacing in Scanning Dir (mm)	Distance between samples. Data points will be collected every time the distance traveled increases by this number.
Sampling Rate (Hz)	Sampling rate desired on laser system. This is used in conjunction with sample spacing to determine feed rates. Fast sampling rates, however, can cause data points to be missed. 1 kHz is recommended sampling speed, but up to 2 kHz is typically reliable.
Setup	Loads the program and verifies that a valid filename and directory have been entered.
Start	Begins scanning.
Stop	Ends scanning and saves data, if only a few degrees of a circular area (R Scan) are desired the program can be stopped after the desired data has been collected.
Data File Setting	Choose directory and filename for data to be saved.
Enable Laser	Must be enabled for data to be collected.

Tool Changer Tab (Automatic Tool Changer Option Required)

The Tool Changer Tab, Figure 4.13, is functional on machines that feature the automatic tool changer system. This tab allows users to interact with the tool changer. There are 36 tool locations in the tool carousel. Table 4.5 lists the applicable buttons and their functions.

Note: The Tool Changer system will home at power up and remain at position “zero” until tool loading begins. It is recommended that all tools are loaded into the wheel and removed from the spindle at power up to ensure proper operation.

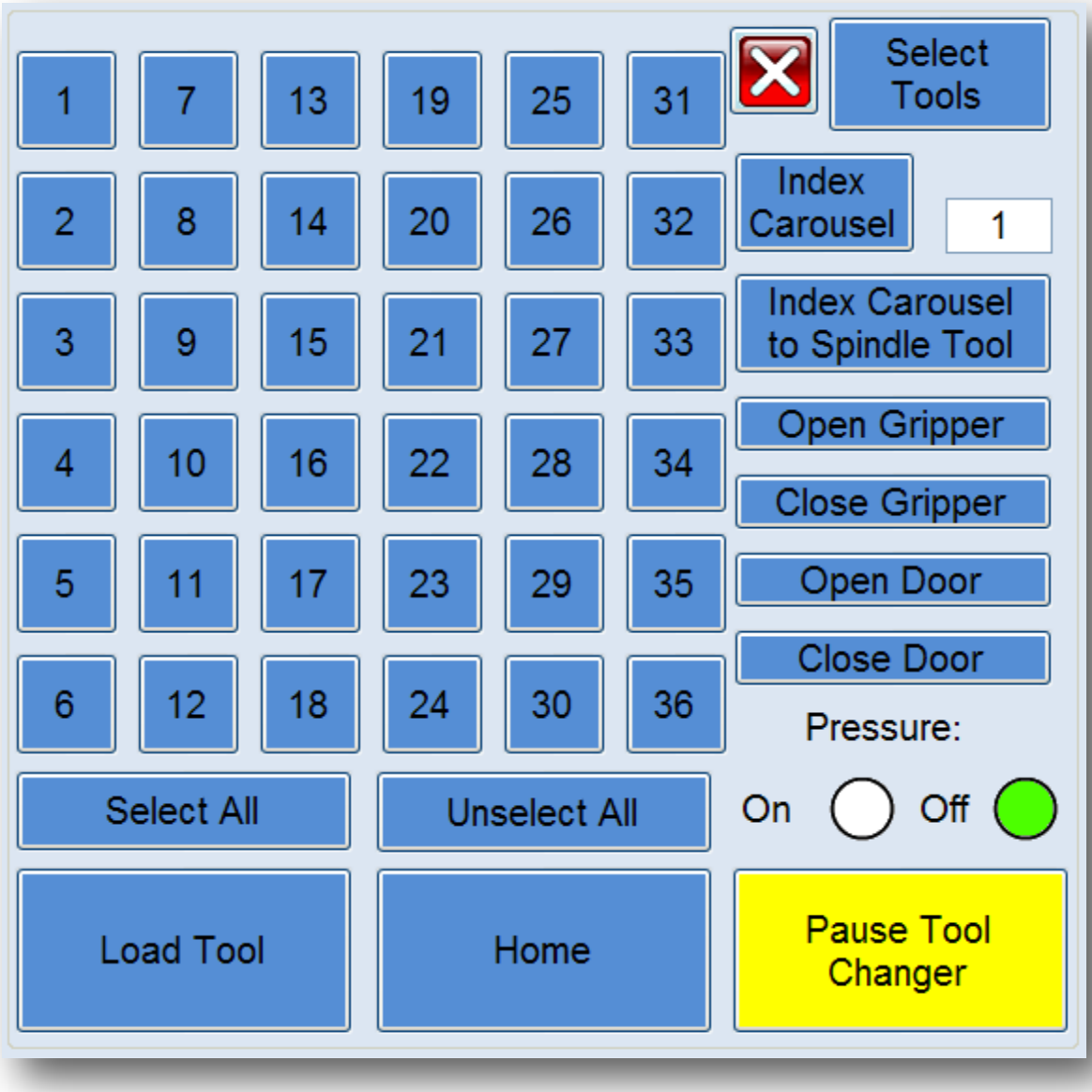


Figure 4.13: Tool Changer Tab

Table 4.5: Tool Changer Tab buttons and their functions

Button	**Function**
Tool Buttons	Shows the user which tools have been selected and are available for tool change. The tool buttons can be highlighted through the *Select Tools* button.
Select Tools	Enables all tool buttons to allow the user to highlight tools that are currently in the system. Pressing the *Select Tools* button a second time will disable all tool buttons and save the current setting.

Index Carousel	Indexes the tool carousel to the tool number listed in the textbox to the right of the button.
Index Carousel to Spindle Tool	Indexes the tool carousel to the tool number that is currently in the spindle, if the spindle tool is known. The spindle tool is established after the first tool change after power up.
Open Gripper	Opens the tool changer gripper to allow user to remove tool if necessary.
Close Gripper	Closes the tool changer gripper to allow user to replace tool if necessary.
Open Door	Opens tool changer door.
Close Door	Closes tool changer door.
Pressure On/Off Switch	Turns the air pressure for the pneumatic tool changer on or off. The air pressure should be on for all tool changes.
Select All	Highlights all tool buttons, indicating that all tools are in the system.
Unselect All	Removes the highlight from all tool buttons, indicating that no tools are in the system.
Load Tool	Performs a tool change. After pressing the *Load Tool* button, choose a highlighted tool button to change to that tool.
"X" Button	Cancels a tool change started with the *Load Tool* button but before the tool is selected.
Home	Resets the tool carousel to ensure that it is in the correct position.
Pause Tool Changer	Pauses the tool change. Can be resumed by pressing the button a second time.

View Panel

Starting the software automatically opens the View Panel as seen in Figure 4.14. View Panel is used for the user to see feedback information from the 363-S mMT and its peripheral devices. The tabs included in this panel are G-Code, Axis, Camera, Laser, Error Status, Warning Status, Tool Offsets, and Workpiece Offsets.

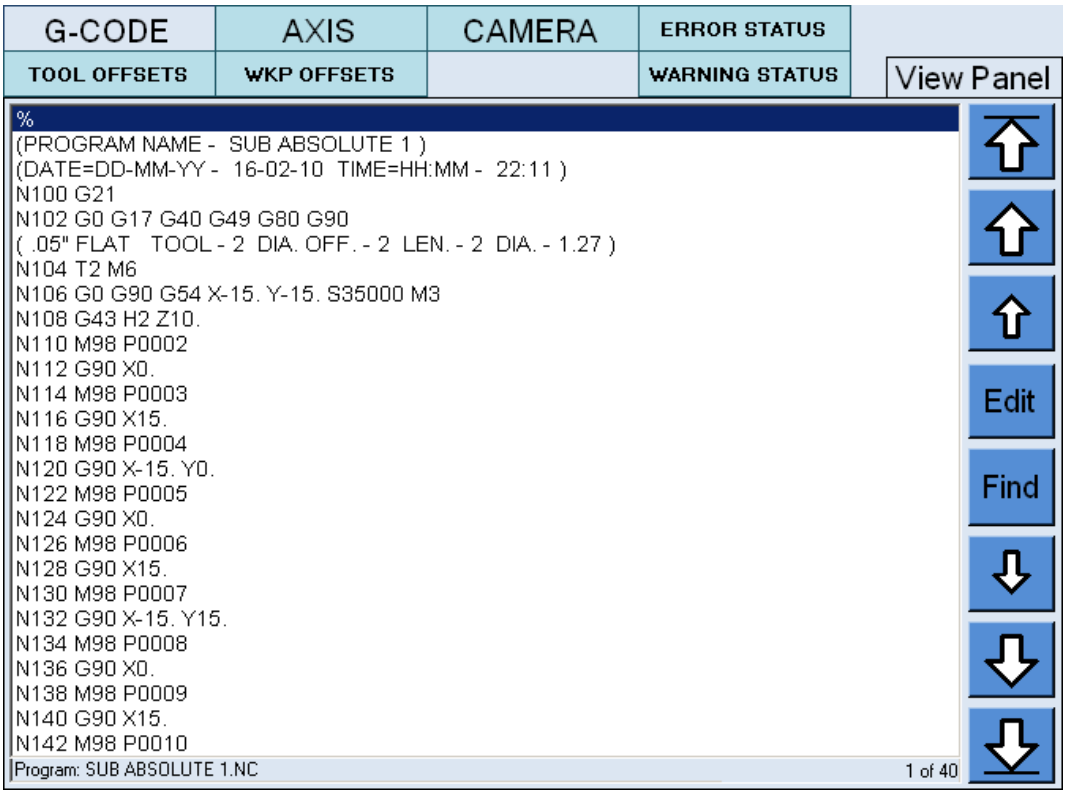


Figure 4.14: View Panel, GCode Tab

G-Code Tab

The *G-Code Tab* in the View Panel (shown in Figure 4.14) displays the current program that is loaded and ready to be run. The program can be searched/scrolled through using the buttons to the right described in Table 4.6. The program can be loaded and unloaded using the *Combine Tab* in the Input Panel. Once a program is successfully loaded, it can be executed with the *Cycle Start* button in the *Program Tab* of the Input Panel. It can be executed from any line in the G-Code window. Note, take care when starting from the middle of the program to ensure all offset, etc are setup correctly. While a program is running, the line of code the program is currently executing is highlighted in the *G-Code Tab*.

Table 4.6: G-Code Window Buttons

Item	Button	Functionality
Begin Button		Moves the G-Code cursor to the beginning of the G-Code program.

Page Up Button		Moves the G-Code cursor up one page of the G-Code program.
Line Up Button		Moves the G-Code cursor up one line of the G-Code program.
Edit Button		Enables the user to edit the currently selected line of G-Code text.
Find Button		Opens a find/replace window allowing the user to quickly search for/replace text. Note, use caution when using the replace all functionality. Replacing many lines in a large file can be very slow.
Line Down Button		Moves the G-Code cursor down one line of the G-Code program.
Page Down Button		Moves the G-Code cursor down one page of the G-Code program.
End Button		Moves the G-Code cursor to the end of the G-Code program.

Axis Tab

The *Axis Tab* in the View Panel opens the Machine Tool Status Graph shown in Figure 4.15. This is a graphical representation of the current position of the cutting tool. The blue lines indicate the program coordinate system. The graph has units of mm and (0,0) refers to the machine origin. The red lines indicate the axes maximums and thus define the working volume of the machine. The user can click, hold, and drag a box on either graph to zoom in. Zooming out is achieved by right clicking and selecting *Un-Zoom* or *Undo All Zoom/Pan*. Right clicking the graphs with the mouse displays such options as copy, save, and print. This feature is useful for monitoring the utilization of the working volume. During setup, this feature can be used to adjust the fixturing of the workpiece so that it falls within the working volume. To verify the workpiece is within the working volume, move the stages [ensure stages are killed (refer to Table 4.5 for more information on killing stages)] around the edges of the workpiece and watch the Machine Tool Status to confirm the workpiece falls within the working volume. The *Clear Part Pattern Array* button will clear any part pattern array that is displayed on the Machine Tool Status Graph.

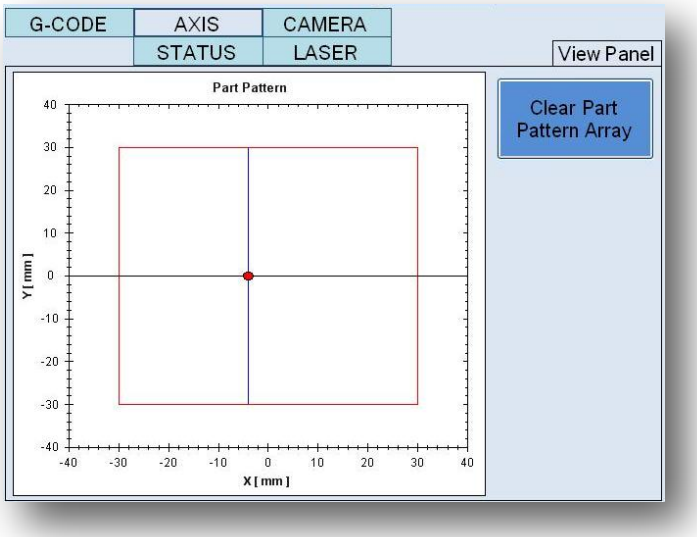


Figure 4.15: Axis Tab showing the Machine Tool Status

CameraTab (Camera Option Required)

The Camera Tab displays the image from an optional camera. This is useful for visual tool settings and other operation monitoring and real time inspection. The contents of this tab will be slightly different depending on the type of camera that is purchased. Fig. 4.16 shows the controls for an EO-3112C Color USB Camera from Edmund Optics. These controls shown here are the same as for the camera's software, so consult the camera's user manual or online help for more information.

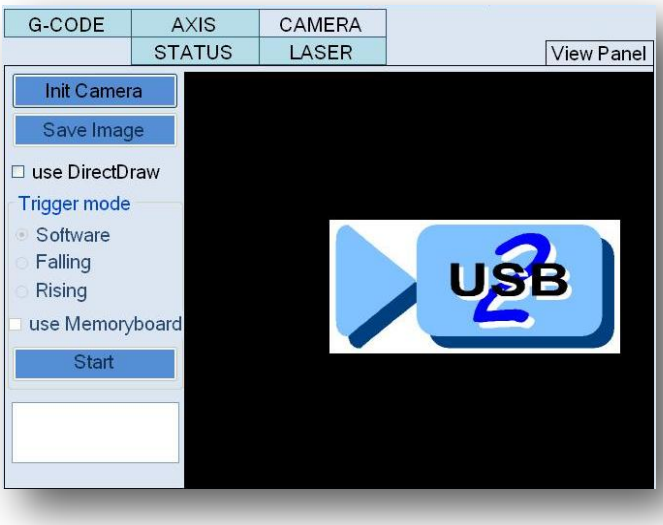


Figure 4.16: Camera Tab

Laser Tab (LT-9010 Confocal Laser Option Required)

The Laser Tab (shown in Figure 4.17) shows the camera feedback from the Keyence LT-9010M or similar laser with a camera in the laser head. This tab will only be available if your laser includes a camera.

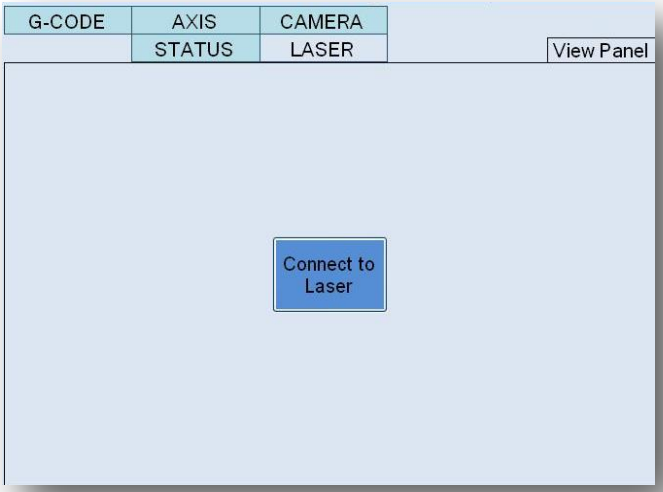


Figure 4.17: Laser tab

Error Status Tab

The Error Status Tab displays the status of error alarms on the 363-S (shown if Figure 4.18). If the indicator next to an alarm is green, then that alarm functioning properly. If the indicator is red, then a fault has been registered and the alarm must be cleared before proceeding with many of the functions on the machine. Table 4.7 gives a description of each of the alarms.

Table 4.7: Error Alarms on the 363-S

Alarm	**Cause(s)**	**System Response**	**Recovery Procedure**
Workpiece Probing Error	See Probing Manual	Machining cycle will stop Spindle rotation will stop	See Probing Manual
Tool Length Measurement Error	Tool measurements not within specified repeatability band Tool sensor beam blocked	Machining cycle will stop Spindle rotation will stop	Check for beam blockage Re-run tool length measurement
Tool Diameter Measurement Error	Tool measurements not within specified repeatability band Tool sensor beam blocked	Program will stop Spindle will stop	Check for beam blockage Re-run tool diameter measurement
Tool Breakage Alarm	Tool breakage detected	System pause	Replace tool

Tool Dirty Alarm	Dirty tool detected	System pause	Clean tool
E-Stop Circuit Tripped	E-stop button pressed Controller watchdog fault Regen resistor temperature fault	Motors de-energized Spindle de-energized	Correct cause of E-stop trigger
X-Axis Travel Limit Exceeded	Software travel limit exceeded	Machine motion stopped	Check for programming errors
Y-Axis Travel Limit Exceeded	Software travel limit exceeded	Machine motion stopped	Jog axes to within travel limits
Z-Axis Travel Limit Exceeded	Software travel limit exceeded	Machine motion stopped	
X-Axis Following Error Y-Axis Following Error Z-Axis Following Error	Following error limit exceeded Following error limit exceeded Following error limit exceeded	All motors de-energized All motors de-energized All motors de-energized	Check for any restrictions to motion Re-initialize machine
X-Axis Amplifier Fault Y-Axis Amplifier Fault Z-Axis Amplifier Fault	Amplifier fault Amplifier fault Amplifier fault	All motors de-energized All motors de-energized All motors de-energized	Re-initialize machine
X-Axis Warning Error Y-Axis Warning Error Z-Axis Warning Error	Warning error limit exceeded Warning error limit exceeded Warning error limit exceeded	All motors de-energized All motors de-energized All motors de-energized	Check for any restrictions to motion Re-initialize machine
Interlock Fault	Attempt to start machining cycle or spindle with lid open	Machining cycle will stop Spindle rotation will stop	Close lid
Tool Changer Fault	Multiple Errors, see software message	Machining cycle will stop Spindle rotation will stop	Follow software message instructions
Positioning Fault	Machine not in home position during home check.	Machining cycle will stop Spindle rotation will stop	Move machine to Home and restart program.
Robot Fault	Issue performing an automatic pallet change.	Machining cycle will stop Spindle rotation will stop	Check for robot issues and restart program.
System Pressure Loss	Loss of system pressure	Machining cycle will stop Machine will move to safe position	Restore system pressure Reinitialize machine

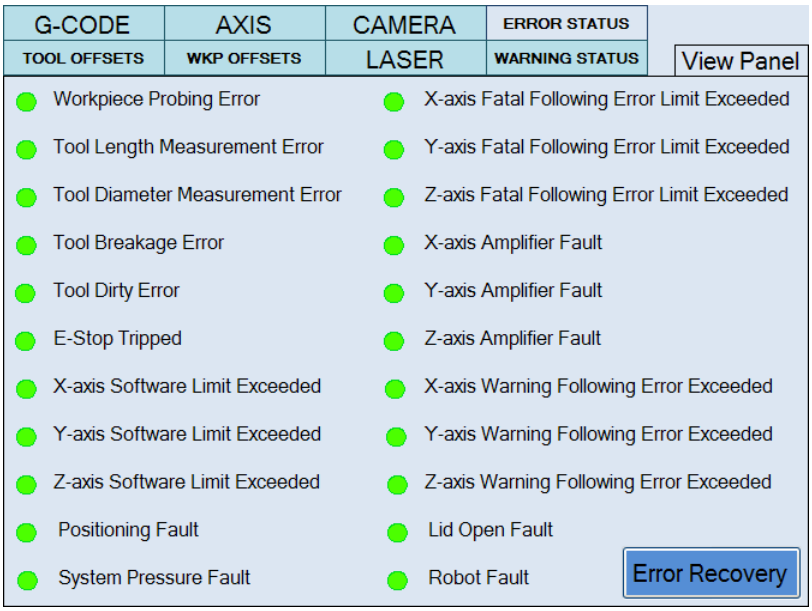


Figure 4.18: Error Status Tab

Warning Status Tab

The Warning Status Tab displays the status of warning alarms on the 363-S (shown if Figure 4.19). If the indicator next to an alarm is green, then that alarm functioning properly. If the indicator is red, then a fault has been registered and the alarm must be cleared before proceeding with many of the functions on the machine. Table 4.8 gives a description of each of the alarms.

Table 4.8: Warning Alarms on the 363-S

Alarm	**Cause(s)**	**System Response**	**Recovery Procedure**
Spindle Motor Disconnected	Spindle motor disconnected	Spindle will not start	Check for loose / disconnected connections
Spindle Warning	High spindle motor cooling air pressure fault Spindle torque overload warning Spindle motor current overload warning Spindle over-speed warning	No action	Check Warning Code on Spindle Controller Correct cause of warning code

Spindle Error	Spindle torque overload fault Spindle motor current overload fault Low spindle motor cooling air pressure fault Spindle over-speed fault Power cycled on controller box with spindle commanded to be spinning	Spindle will stop Program will stop Spindle will not start	Check Error Code on Spindle Controller Correct cause of error code
X-Axis Velocity Error Y-Axis Velocity Error Z-Axis Velocity Error	Velocity limit exceeded Velocity limit exceeded Velocity limit exceeded	Machining cycle will stop Spindle rotation will stop	Re-initialize machine

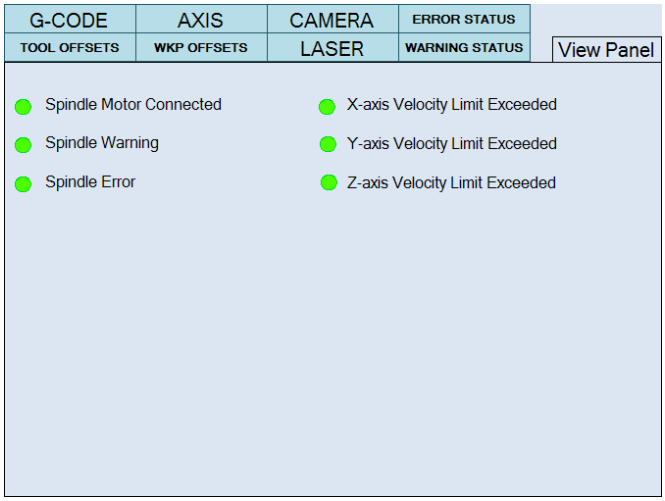


Figure 4.19: Warning Status Tab

Workpiece Offsets Tab

The Workpiece Offsets Tab displays the values of all the workpiece offsets on the 363-S (shown if Figure 4.20). All offsets can be accessed through paging back and forth or through searching for an offset number. Offsets can be saved and recalled for later use. Table 4.9 gives a description of each of the items on the panel.

Table 4.9: Workpiece Offsets on the 363-S

Item	**Functionality**
Previous Page Button	The previous page button allows users to flip back through the previous pages of offsets stopping at the first page.
Index to Offset Button	The index to offset button enables the user to type in an offset number, press the button and be automatically taken to the correct offset page with the desired offset selected.

Next Page Button	The next page button allows users to flip through the next pages of offsets stopping at the last page.
Workpiece Offset Table	The workpiece offset table holds the offsets for G-Codes G54 – G59 and G54.1 P1 – 394. These offsets can be updated by the user through this table.
Save to File Button	The save to file button writes all the current workpiece offsets to a file and saves it to a user specified destination.
Retrieve from File Button	The receive from file button takes a user specified offset file and replaces all the current workpiece offset with the offsets from that file.

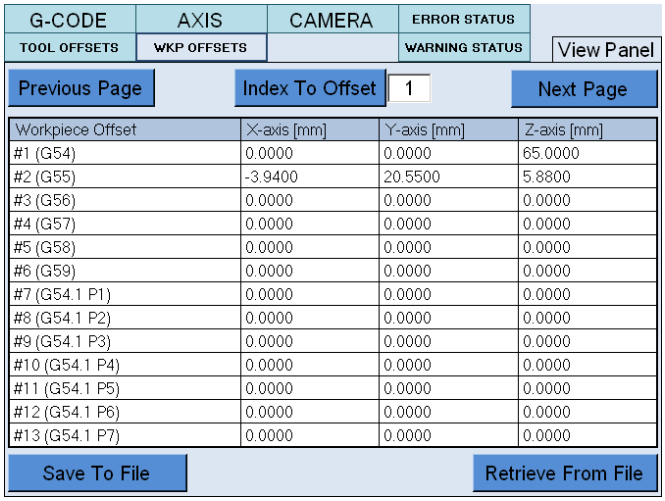


Figure 4.20: Workpiece Offsets Tab

Tool Offsets Tab

The Tool Offsets Tab displays the values of all the tool offsets and compensation factors on the 363-S (shown if Figure 4.21). All offsets can be accessed through paging back and forth or through searching for an offset number. Offsets can be saved and recalled for later use. Table 4.10 gives a description of each of the items on the panel.

Table 4.10: Tool Offsets on the 363-S

Item	**Functionality**
Previous Page Button	The previous page button allows users to flip back through the previous pages of offsets stopping at the first page.
Index to Offset Button	The index to offset button enables the user to type in a tool offset number, press the button and be automatically taken to the correct offset page with the desired offset selected.
Next Page Button	The next page button allows users to flip through the next pages of offsets stopping at the last page.
Tool Offset Table	The tool offset table holds the important information about each tool including tool position, description, length measurement, diameter measurement, length compensation and diameter compensation. The description, length compensation and diameter

	compensation can be updated by the user.
Save to File Button	The save to file button writes all the current tool offsets to a file and saves it to a user specified destination.
Retrieve from File Button	The receive from file button takes a user specified tool offset file and replaces all the current tool offsets with the offsets from that file.

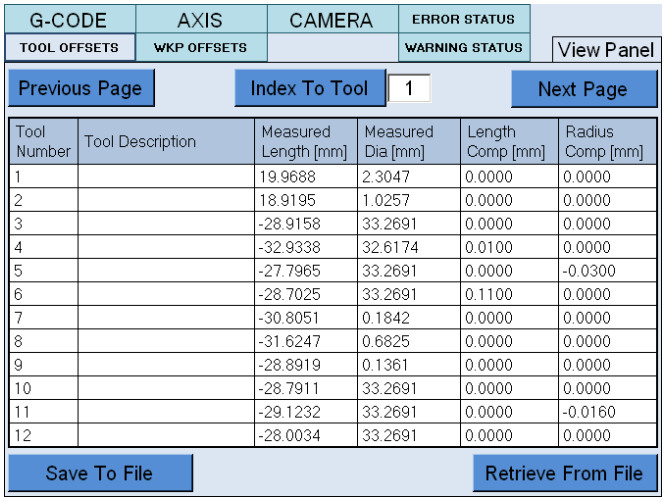


Figure 4.21: Tool Offsets Tab

Input Panel

Starting the software automatically opens the Input Panel as seen in Figure 4.22. The Input Panel is used to perform manual operations, program loading, and direct interaction with the machine controller.

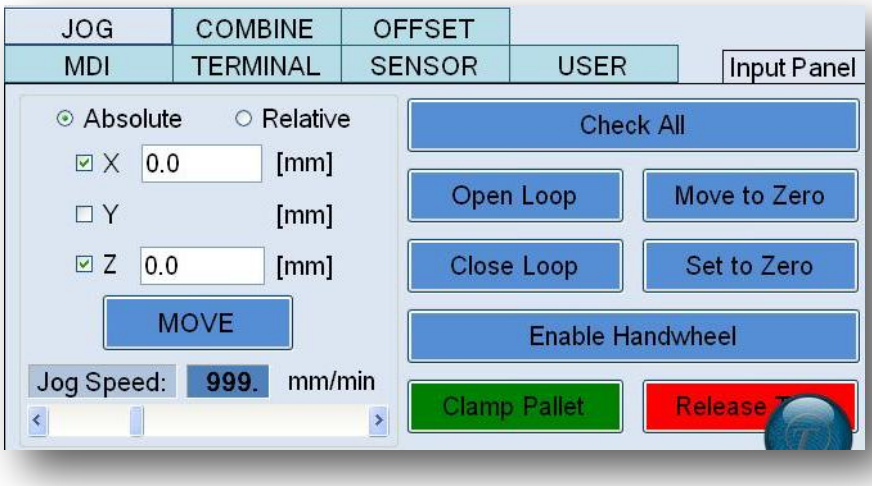


Figure 4.22: Input Panel displaying the Jog Tab

Jog Tab

The Jog Tab is used for manually moving the axes and loading / releasing tools manually. The axes can be moved incrementally or to absolute positions, and the handheld pendant can also be enabled to manually move the axes. Figure 4.22 shows the Jog Tab and Table 4.11 lists the buttons and their functions. Refer to the Operation section of this manual for a step – by – step set of instructions for using the Jog Tab.

Table 4.11: Jog Tab buttons and their functions

Button	**Function**
Move	Use the Move button to move the stages to absolute or relative coordinates according to the values specified by the text boxes. Only the axes that are checked will be moved.
Jog Speed	The jog speed text box displays the current jog speed for all three axes. Moving the slider bar will adjust the jog speed accordingly.
Check All	Apply action to all axes.
Clamp/Unclamp Pallet	Releases and clamps the pallet. Pressing this button when it displays *Clamp Pallet* will clamp the pallet. Pressing the button when it says *Unclamp Pallet* will unclamp the pallet.
Clamp/Release Tool	Releases and clamps the automatic tool chuck (automatic tool changing spindles only). Pressing this button when it displays *Release Tool* will release the tool. Pressing the button when it says *Clamp Tool* will clamp the tool.
Move to Zero	Moves to program origin.
Set to Zero	Sets current axis position to program zero.
Close Loop	Set selected axes to closed loop (enable). The motors will keep the axis in the commanded position when in closed loop mode.
Open Loop	Set selected axes to open loop (disable). When an axis is in open loop mode it can be moved manually.
Enable Handwheel	Pressing the button when it displays *Enable Handwheel* will enable the handwheel on the handheld pendant. Pressing the button when it displays *Disable Handwheel* will disable the handwheel.

MDI Tab

The MDI (Manual Data Input) Tab is used for manually executing G- and M-code commands outside of a CNC-program (shown in figure 4.23). Commands can be executed by typing them into the text box. Multiple lines can be executed at the same time by typing them all in the text box and pressing the *Execute* button. There are buttons for most G- and M-code commands that are available on the 363-S. Pressing one of these buttons will automatically insert the command into the text box.

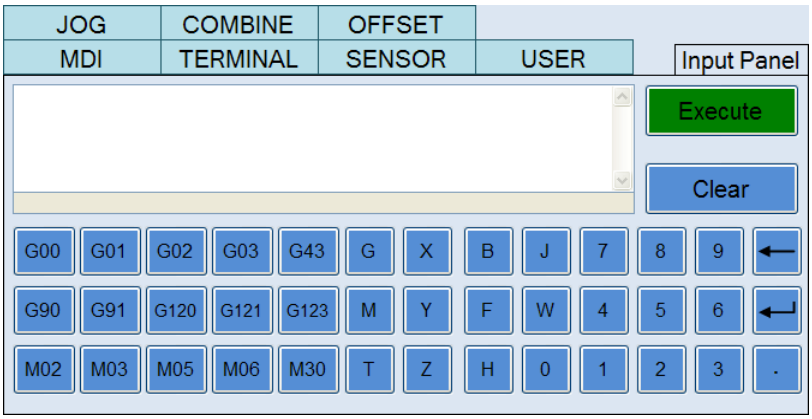


Figure 4.23: MDI Tab

Combine Tab

The Combine Tab is used to find, queue, load, and rewind G-Code files. The Combine Tab is located in the Input Panel as shown in Figure 4.24 below. Table 4.12 below lists the buttons and functions of the Combine Tab.

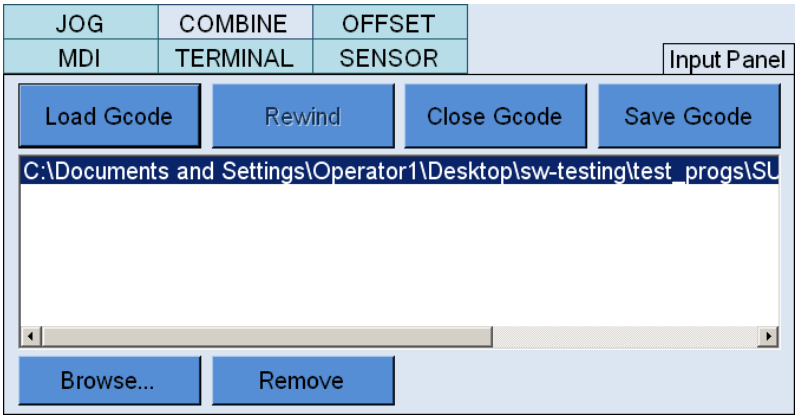


Figure 4.24: Combine Tab

Table 4.12: Combine Tab buttons

Button	**Function**
Load Gcode	Loads all the G-Code files from the queue to memory and displays the code in the G-Code text box of the View Panel. Also, downloads all subprograms.
Rewind	Makes the first line of the program active. This can be used after a pause command or stop command. The program will not begin executing until Cycle Start is pressed.
Close Gcode	Removes the current G-code program from memory and clears the text box displaying the code.
Save Gcode	Takes the currently displayed Gcode and save it to a file of the user's choosing.

Browse…	Browse and find G-Code files. After finding a file, it will show up in the queue box.
Remove	Removes the selected G-Code file from the queue.

Terminal Tab

The Terminal Tab located on the Input Panel opens a Terminal which allows commands to be sent to the controller one line at a time. Figure 4.25 shows the Terminal. Commands are entered in the text box at the bottom of the window and sent with the enter key.

Warning: The Terminal Panel has the ability to directly address variables and functions in the Delta-Tau CNC controller and therefore should be used with caution. Changes to key operating variables can result in changes in machine accuracy, erratic machine behavior, damage to the machine and/or personal injury. Only personnel specially trained in the operation of the Terminal Panel functionality should use any commands in the Terminal Panel other than the "run" command. The "run" command executes a program. To use the "run" command, type b###r into the Terminal Panel, where ### is the program number. For example, typing b300r would run program 300. Damage to the machine caused by misuse of the Terminal Panel may void the factory warranty.

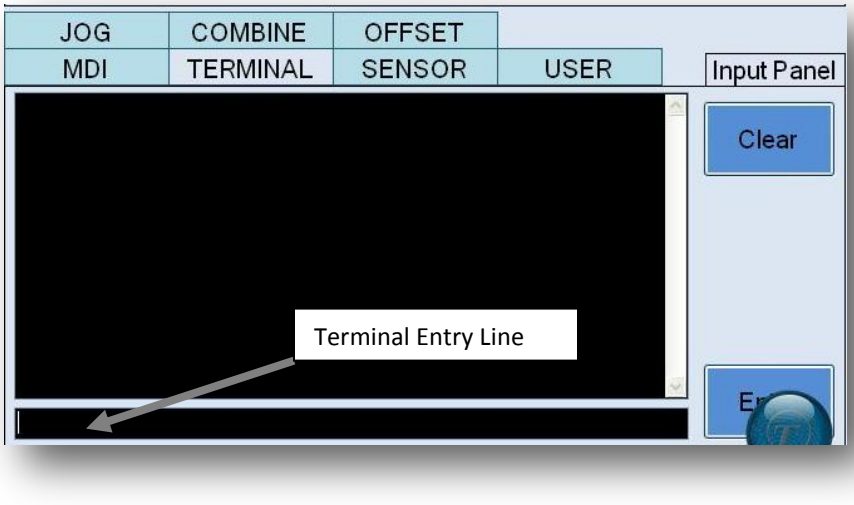


Figure 4.25: Terminal Tab

Offset Tab

The Offset Tab (see Figure 4.26) is used to set, save, and load program origins (workpiece offsets). Table 4.13 describes the buttons located within the Offset Tab and their functions. Refer to the Operation section of this manual for a step – by – step set of instructions for using the Offset Tab.

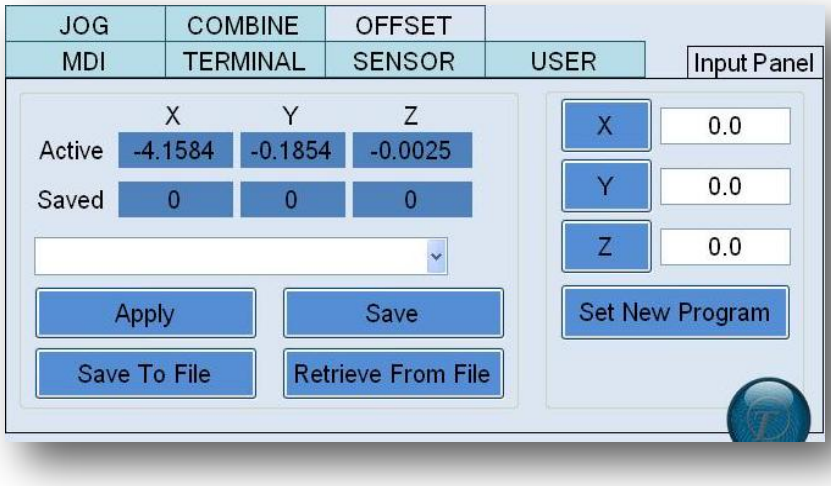


Figure 4.26: Offset Tab

Table 4.13: Offset Tab buttons and their functions

Button	**Function**
Apply Saved	Applies a previously saved program origin as the current program origin.
Save Active Origin	First, select the desired offset (G54 through G59) from the drop down box. Next, the *Save Active Origin* button will save the active origin as the selected offset.
Save To File	The *Save to File* button will save all of the offsets (G54 through G59) to a user selected file. This allows multiple origin offsets to be saved for later use.
Retrieve From File	Retrieves a set of saved program offsets (G54 through G59) from the saved file.
Set New Program	Updates the active program origin with the values next to the selected X, Y, or Z buttons.
X	If enabled (Orange), the active origin will be updated with the number next to this button when the *Set New Program* button is pressed.

Y	If enabled (Orange), the active origin will be updated with the number next to this button when the *Set New Program* button is pressed.
Z	If enabled (Orange), the active origin will be updated with the number next to this button when the *Set New Program* button is pressed.

Sensor Tab (Tool Sensor option required)

The Sensor Tab (shown in Figure 4.27) is used to perform an automated tool sense routine. There are buttons for both a tool length sense and a tool diameter sense. When the operations are finished, the result data is displayed in the adjacent text box. This data is automatically saved into the registers for the tool number listed in the textbox in the top-right. No experience value is used when performing a tool sense with this method. The *Select Carousel Tools* button enables the user to detect which tools are present in the tool carousel. The *Sense Selected Tools* button uses the highlighted tools in the Tool Changer Tab to change to and sense each tool. See the Tool Sensing section of this manual for more information about the tool sensing routine. If the workpiece probing option is available on the machine, after a pallet identification routine, the pallet number will be displayed in the *Pallet ID* textbox. See the Workpiece Probing manual for more information about the workpiece probing.

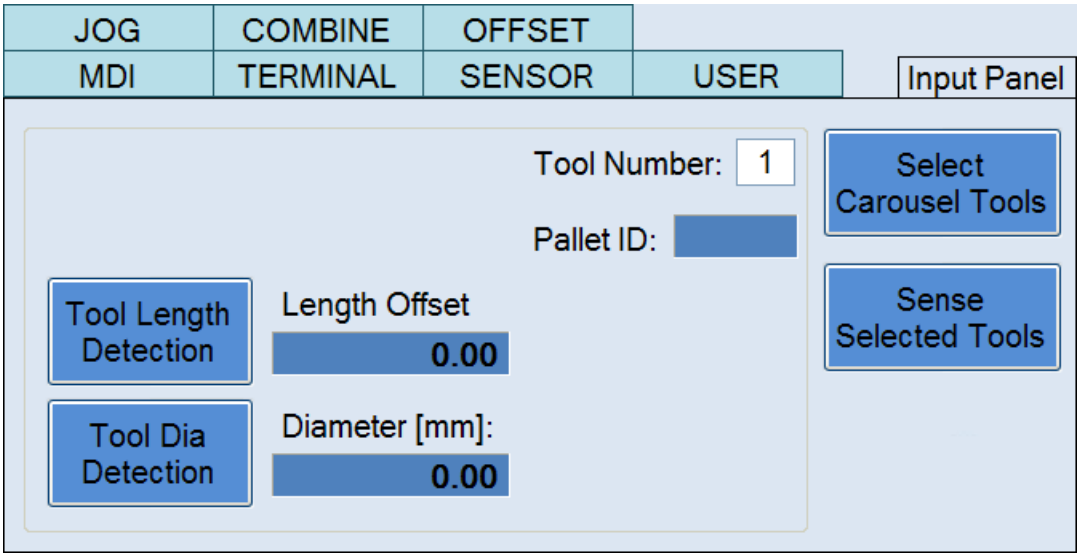


Figure 4.27: Sensor Tab

User Tab (Options required)

The User Tab, Figure 4.28, may vary depending on special options and equipment included on individual machines. Typical options found here will be program array setup and manual operation of special pneumatic systems. These system include tool guarding, mandrel clamping, and robot door actuation.

The program array setup can be used to repeat a loaded program at each spot on the array. These locations are defined in the workpiece offset table, offsets #1 - #12. Select the correct pockets for milling using the Set Pockets button, browse to the correct program file through the Browse button, and then load the program by clicking the Load Program button. Press Cycle Start to begin.

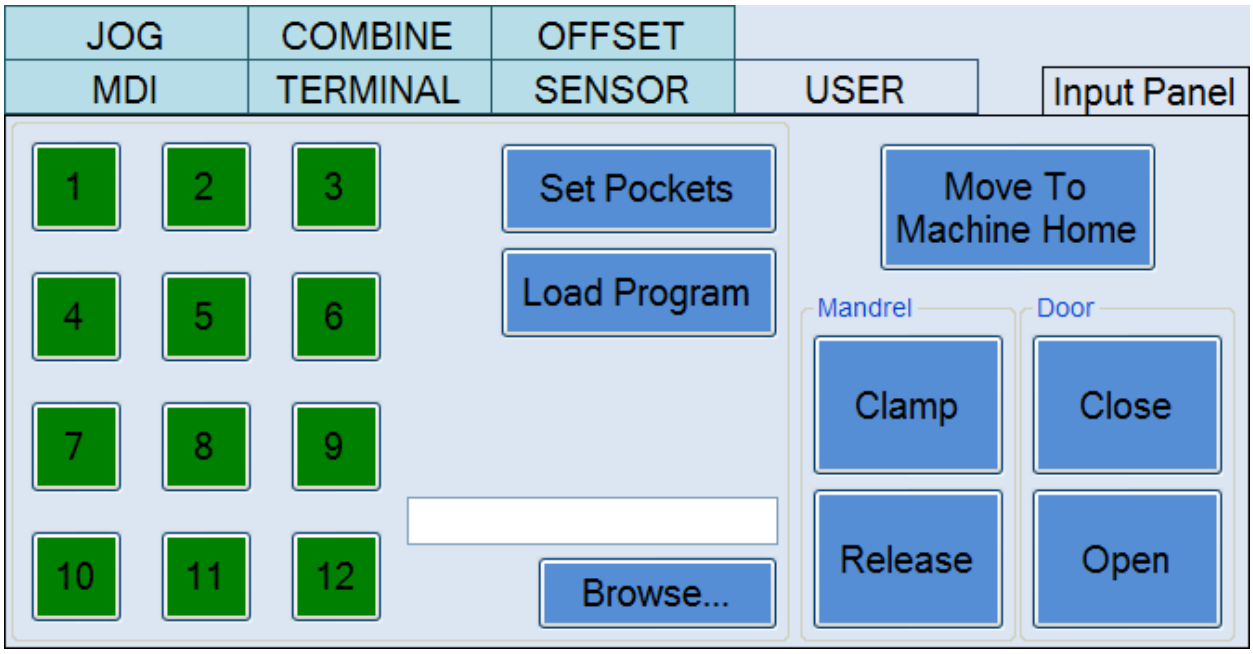


Figure 4.28: User Tab

HARDWARE INTRODUCTION

Coordinate Axes

The coordinate system for the 363-S is a right-handed system set up with the positive x-axis extending to the right (when facing the machine), the positive y-axis extending up, and the positive z-axis extending toward the front of the machine as seen in Figure 5.1.

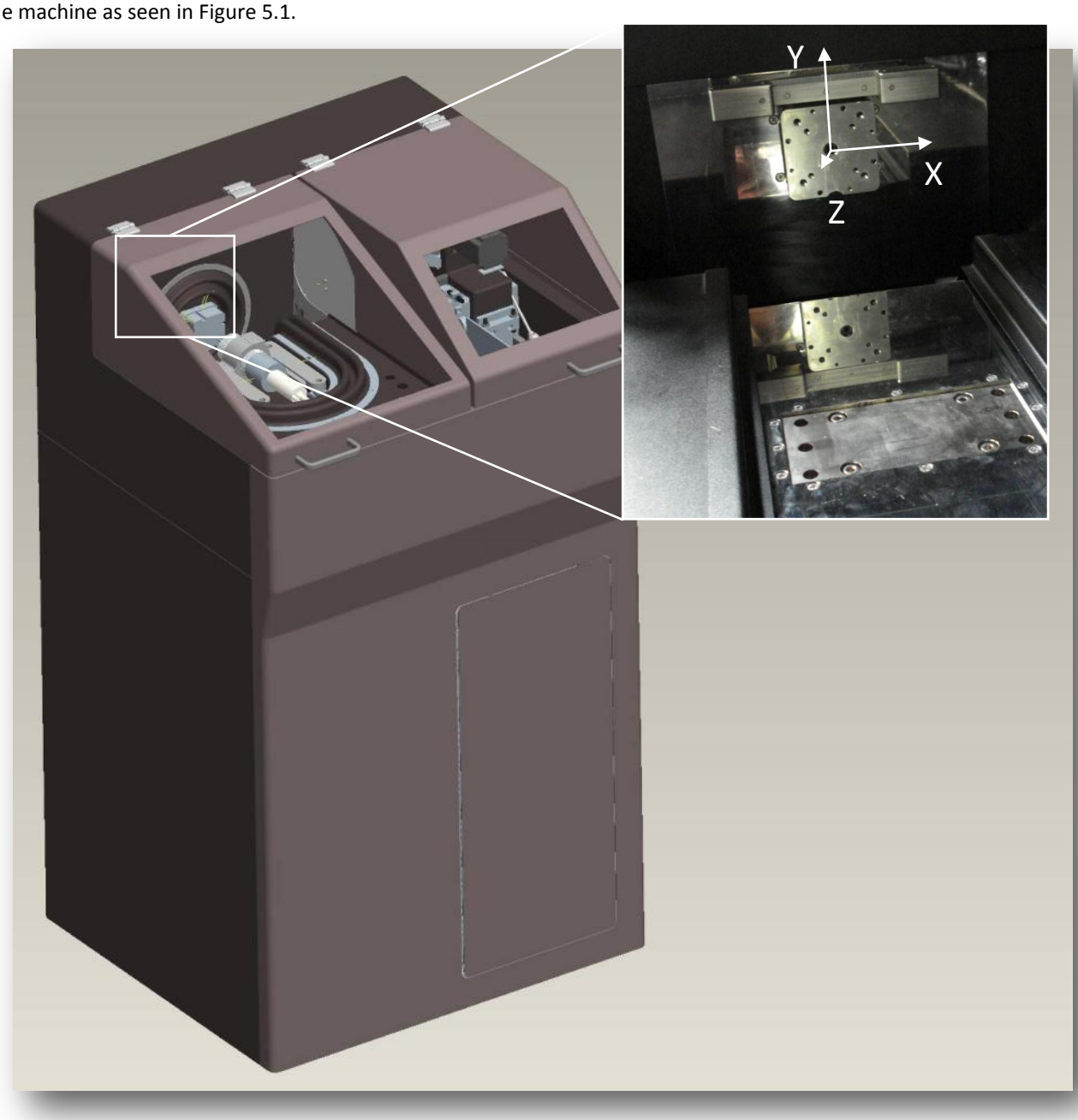


Figure 5.1: Coordinate axes

Pallet and Spindle Mounting

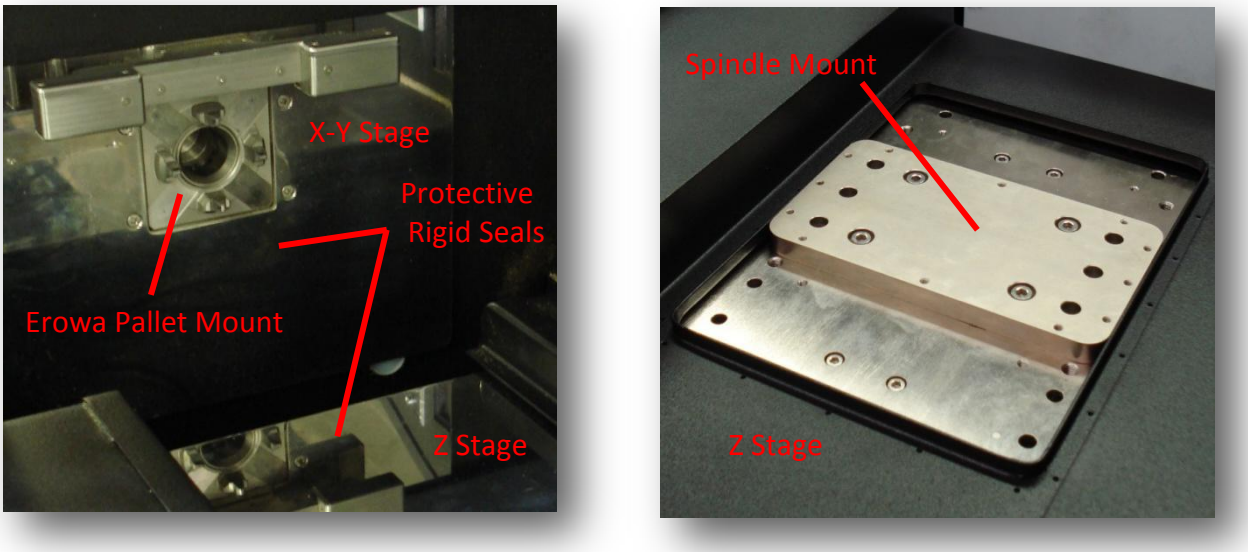


Figure 5.2: X-Y-Z stage introduction

The following items reference Figure 5.2.

X-Y Stage
The X-Y stage moves in the X-Y plane and has 2.48" (63 mm) of travel in each direction. Pallets (see "Work Piece Mounting System" Section for an introduction to pallets) are mounted to the Erowa pallet mounting system located on the X-Y stage.

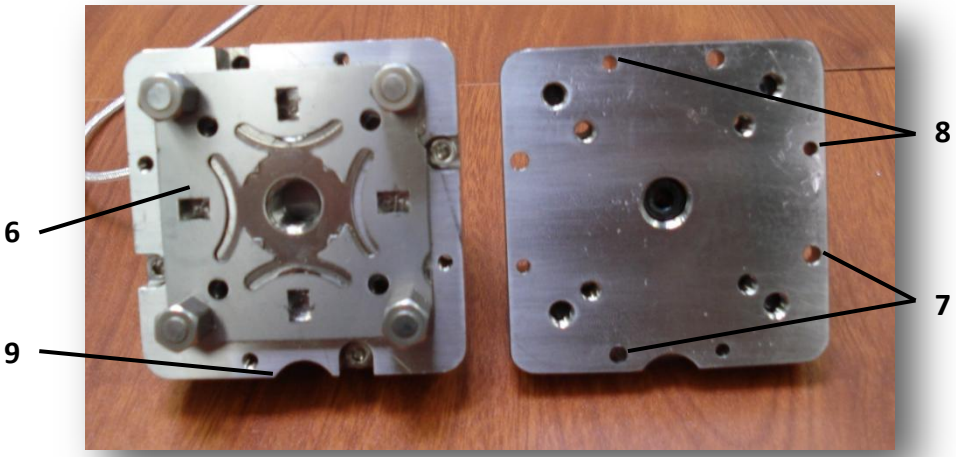
Erowa Pallet Mounting System
The Erowa pallet mount system allows pallets to quickly and easily be changed with sub-micron repositioning repeatability. The mounting system uses a pneumatic chuck to position the pallet. This connection assures repeatable positioning.

Protective Rigid Seals
The protective rigid seals keep chips and fluid out of the machine motion and electronic components while still allowing motion of the stages.

Z Stage
The Z stage moves in the Z direction and has a total travel range of 3.94" (100 mm). Available machining travel range depends on the spindle, workpiece fixture, and installed options (such as an automated tool changer). Spindles (detailed in the Spindle Introduction) and optional metrology equipment are mounted to the spindle mounts located on the Z stage.

Spindle Mount
The spindle mount allows for quick spindle changes with highly repeatable positioning. The spindle mounting system uses 6 bolts to rigidly hold the spindle block to the z-axis stage.

Work Piece Mounting System



1. Pallet Back and Front

2. Spigot

3. Pallet with Spigot Attached

4. Machinable Fixture Plate

5. Complete Pallet System

The following items reference Figure 5.3. Figure 5.3: Pallet System Introduction

1. Pallet

The front face of the pallet is smooth with four threaded (7) and four unthreaded (8) mounting holes around the edges of the pallet. The mounting holes enable the machinable fixture to be securely fastened to the pallet. The notch (9) seen at the bottom of the pallet indicates the correct mounting orientation. The notch should face towards the floor when mounted to the machine. The backside of the pallet has the Erowa pallet mounting system. This system permits easy and repeatable mounting of the pallet/workpiece to the X-Y stage.

6. Erowa Pallet Mounting System

The Erowa pallet mounting plate ensures accurate and repeatable pallet placement on the X-Y stage.

7. Threaded Mounting Holes

These four holes allow the interface plate and/or the machinable fixture to be securely mounted to the pallet.

8. Non-Threaded Mounting Holes

The non-threaded mounting holes allow for quick and secure mounting of the machinable fixture plate to the pallet.

9. Notch

There is a notch in the pallet and machinable fixture plate to indicate the correct mounting orientation (notches should be aligned). When mounted on the machine, the notch should face towards the floor.

2. Spigot

The spigot rigidly holds the pallet to the X-Y stage. The Erowa plate on the back of the pallet ensures that the spigot positioning is very accurate.

3. Pallet with Spigot Attached

This shows the correct orientation of the spigot and the pallet.

4. Machinable Fixture Plate

The aluminum machinable fixture plate offers a smooth surface upon which to mount the workpiece. The plate also functions as a sacrificial surface for machining a through feature.

5. Complete Pallet System

Picture five shows the correct completed pallet system.

Spindle Introduction

There are several different spindle options available for the Microlution 363-S. These options include: 160 000 RPM air turbine-air bearing spindle (ABS-1600), 80 000 RPM electric motor driven spindles (NRA-5080), and 40 000 (NR-5041E ATC) and 50 000 (NR40-5100 ATC) RPM electric motor driven spindles with automatic tool changing functionality.

40 000 (NR-5041E ATC) & 50 000 (NR40-5100 ATC) RPM Electric Motor Driven Spindles with Automatic Tool Changing Capability (pictured in Figure 5.4)

Ideal for larger Diameter tools up to 0.125" (3.175 mm)
These spindles are driven by an electric brushless motor. The brushless motor provides a long service life, excellent durability, and high reliability. Both the 40 000 and 50 000 RPM spindles utilize an air operated collet to easily change tools. Both spindles use high precision bearings to obtain spindle accuracy within 2 microns.

50 000 (IBAG - HF45A60CP) RPM Electric Motor Driven Spindles with Automatic Tool Changing Capability

Ideal for larger Diameter tools up to 0.276" (7.0 mm)
These spindles are driven by an electric brushless motor. The brushless motor provides a long service life, excellent durability, and high reliability. This spindle utilizes an air operated collet to easily change tools and high precision bearings to obtain spindle accuracy within 2 microns. The spindle can be operated at a maximum RPM of 60,000 if a liquid chilled spindle block is used to house the spindle. Please contact Microlution for more details.

100 000 (Jaeger: Z33-D11100.02S2Y) RPM Ceramic Hybrid Ball Bearing Spindle with Automatic Tool Changing Capability

Ideal for larger Diameter tools up to 0.125" (3.175 mm)
These spindles are driven by an electric brushless motor. The motor/spindle combination is connected to a spindle control unit which controls the spindle speed. The brushless motor provides a long service life, excellent durability, and high reliability. This spindle utilizes an air operated collet to easily change tools and high precision bearings to obtain spindle accuracy within 2 microns. Please contact Microlution for more details.

80 000 RPM Electric Motor Driven Spindles (NRA-5080)

Ideal for use with tools up to: 0.050" (1.27 mm) in diameter for the air bearing option and 0.125" (3.175 mm) in diameter for the ceramic bearing option
These spindles use an electric brushless motor to drive different spindles. The brushless motor provides a long service life, excellent durability, and high reliability. The motor/spindle combination is connected to a spindle control unit which controls the spindle speed. There are two different bearing options for the 80 000 RPM Electric Motor Driven Spindle. These two bearing options are air bearings and ceramic bearings. The air bearing spindle produces extremely low heat and vibration and has a very long service life because of low friction and wear. The air bearing provides exceptional spindle accuracy of within one micron. The ceramic bearing spindle has a spindle accuracy within one micron and requires an oil mist air flow for lubrication.

160 000 RPM Air Turbine-Air Bearing Spindle (ABS-1600)

Ideal for use with small tools up to 0.02” (0.508 mm) in diameter

Air turbine spindles use an air turbine instead of an electric motor to drive the spindle. The air turbine-air bearing spindle has a maximum spindle speed of 160 000 rpm which is much higher than electric motor driven spindles. Additionally the air turbine-air bearing spindle uses an air bearing which produces extremely low heat and vibration and has a very long service life because of low friction and wear. The air bearing provides a spindle accuracy within one micron.

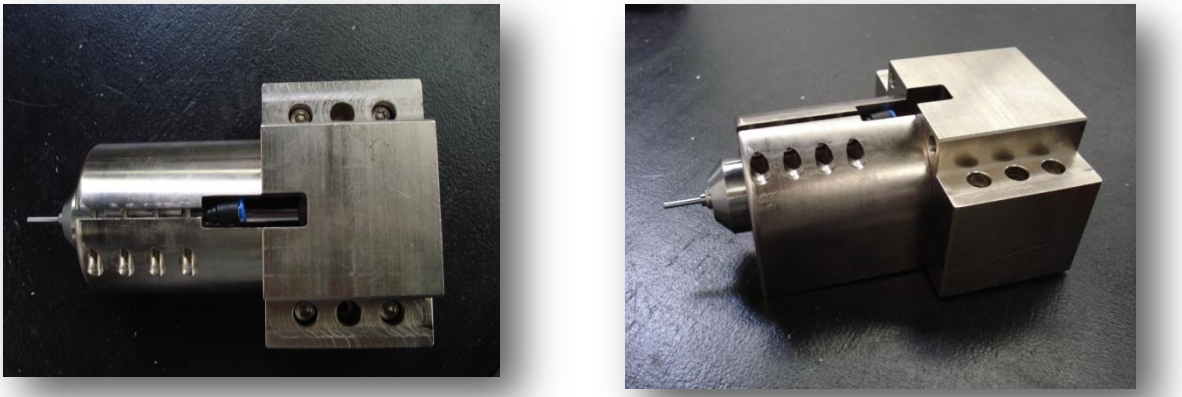


Figure 5.4: One spindle option, 50 000 RPM Electric Motor Driven
Automatic Tool Changing Spindle, with mounting system

Front of the Machine

Figure 5.5 displays the front of the 363-S. The protective lid encloses the machining process and protects the user from chips, coolant, and the moving parts. The protective lid also keeps chips and coolant from exiting the working area. The computer monitor functions as the display for all machining software. The machine body and control systems housing protects the control systems and shields users from the internal electronics. The vibration isolation pads shelter the machine from any floor vibrations. The spindle control unit manages the spindle speed and monitors any spindle errors. Refer to the Spindle Control Unit section for detailed information about the spindle control unit. The handheld pendant allows the user to manually control the movement of the X, Y, and Z stages. The emergency stop button is located on the handheld pendant. More information about the handheld pendant is available in the Handheld Pendant section and in the Using the Handwheel section.

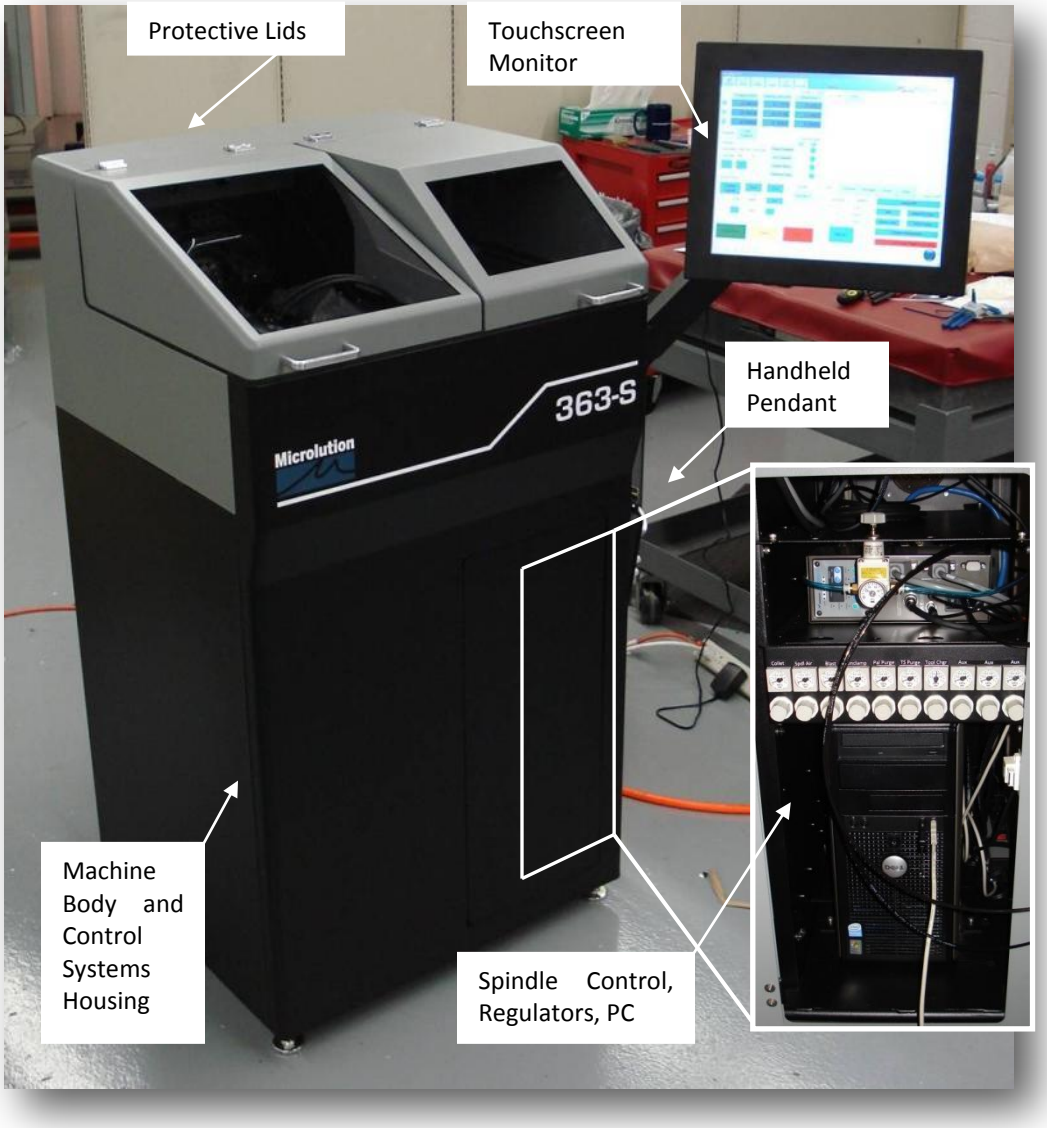


Figure 5.5: Front of the machine

Handheld Pendant

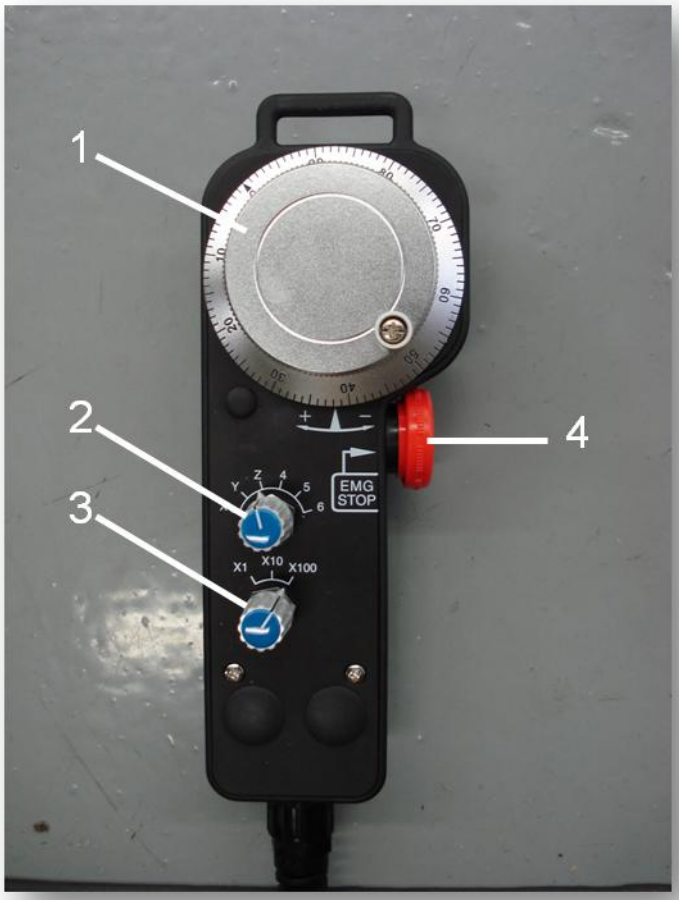


Figure 5.6: Handheld pendant

1. Handwheel
   Each rotation increment of the handwheel moves the selected axis by the selected increment amount.
2. Axis Selection Dial
   Adjusts which axis the handwheel controls. (Positions 4, 5, and 6 are currently inactive).
3. Increment Selection Dial
   Adjusts the handwheel scaling factor. X1 moves each stage one micron per rotation increment, X10 moves each stage ten microns per rotation increment, and X100 moves each stage 100 microns per rotation increment.
4. Emergency Stop Button
   The Emergency stop button shuts off the power to the spindle and stage motors.

Note: To return the machine to normal operation after pressing the E-stop, twist the E-Stop clockwise to disengage it. Next, the linear motors need to be in closed loop mode (Refer to the MDI tab segment of this manual for more information about the *Close* button and closed loop mode.) The spindle must also be restarted. The machine is ready to continue normal operation. Enable the spindle within the Program Run tab and click the *Cycle Start* button to start machining.

Machine Regulators

CB. X-Y stage Counterbalance Pressure Regulator
Suspends the X-Y stage to decrease the workload of the Y-axis linear motors. Pressure will vary based on the weight on the workpiece. Adjust the pressure until the X-Y stage is suspended and stationary.

1. Regulator 1: Tool Collet Pressure Regulator / Airbearing Spindle (optional)
   Regulates the pressure used during tool changes on automatic tool changing spindles. Aux 1 can also be used to supply air to the air bearing spindle options. Suggested air pressure is approximately 0.6 MPa.

2. Regulator 2: Spindle Cooling Air Pressure Regulator
   Regulates the spindle motor cooling air pressure. Suggested air pressure is approximately 0.3 MPa.

3. Regulator 3: Coolant Blast Air Regulator
   Regulates coolant air applied directly to the work piece. Suggested air pressure is approximately 0.2 MPa.

4. Regulator 4: Pallet Unclamp Regulator
   Regulates air pressure of kinematic pallet unclamp action. Suggested air pressure is approximately 0.55 MPa.

5. Regulator 5: Pallet Air Purge Regulator
   Regulates air pressure of pallet cleaning air purge. Suggested air pressure is approximately 0.5 MPa.

6. Regulator 6: Tool Sensor Purge Air (option required)
   Regulates air pressure of tool sensor purge. Suggested air pressure is approximately 0.4 MPa.

7. Regulator 7: Tool Changer Air Regulator (option required)
   Regulates air to the pneumatic actuators on the tool changer. Suggested air pressure is approximately 0.6 MPa.

8. Regulator 8: Tool Clamp Air Regulator (option required)
   Regulates air to provide clamping force to the spindle collet (HF45 spindle option required). Suggested air pressure is 0.2 MPa.

9. Regulator 9: Configurable Auxiliary Regulator
   Contact Microlution.

10. Regulator 10: Configurable Auxiliary Regulator
    Contact Microlution.

Note: Regulators may have different functions depending on the options chosen for the machine. Please contact Microlution for the correct regulator function map if yours do not match what is shown above.

Figure 5.8: Pneumatic regulators

Back of the Machine

The back of the machine consists of all the air, power, computer, and accessory cable connections. Figure 5.9 shows the back of the machine as well as all of the cables connection locations.

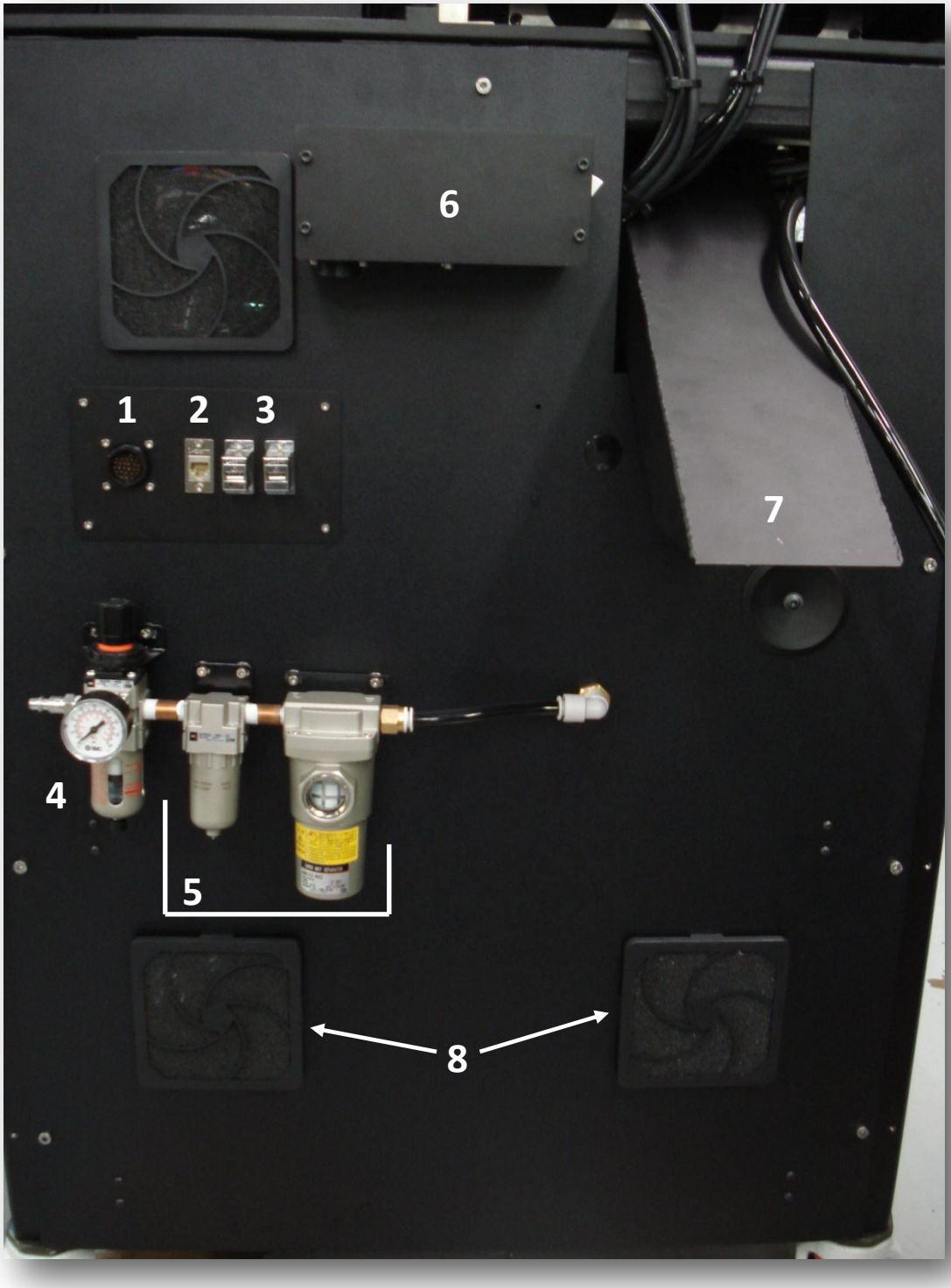


Figure 5.9: Back of the machine

The following items reference Figure 5.9.

1. Handheld Pendant Connection
2. Ethernet Connection
3. USB Connection (2 ports)
4. Compressed Air Input and Regulator (air input is similar to McMaster p/n 6534K46)
5. Air Filter and Mist Separator
6. Power Entry Box
7. Chip Chute
8. Electrical Cabinet Cooling Fans

INSTALLATION / TRANSPORTATION

Installation

Intstallation must be performed by Microlution technician. The steps below are provided for customer reference. The 363-S should be stored and operated according to specifications listed in Table 3.3. Air and power should meet specifications listed in Table 3.2.

The following installation instructions reference Figure 6.1.

1. Place the supplied isolation pads under the feet of the 363-S.
2. Use a wrench to adjust the feet height until the machine is level.
3. Make sure all equipment is switched off.
4. Attach air supply to inlet on back of machine (as shown in Figure 6.1).
5. Attach power cable to rear of 363-S, but DO NOT plug into 120 VAC outlet. (see Figure 6.1)
6. Connect the monitor to the machine (see Figure 6.1).
7. Attach handheld pendant to rear of 363-S (see Figure 6.1).
8. Attach manual laser pendant to rear of 363-S (if equipped).
9. Verify filters for coolant and air are in place.
10. Add coolant to filter case/reservoir until the coolant level is one to two inches below the top of the filter.

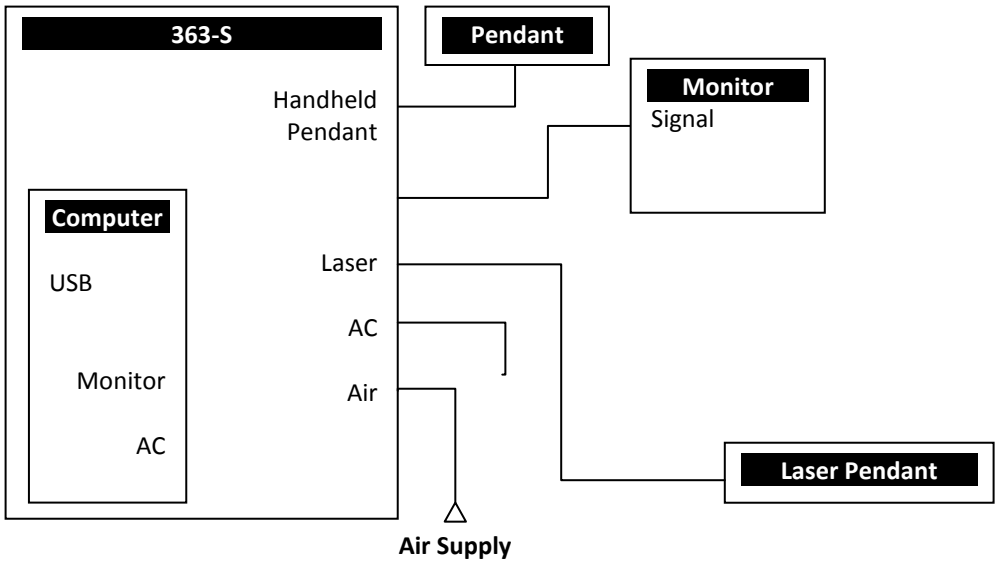


Figure 6.1: Machine air, electric, and peripheral connection schematic

Transportation

The 363-S should be moved with a pallet jack or fork lift.

Warning: The 363-S weighs more than 317 kg [700lbs] and can cause serious injury or death if handled improperly.

Warning: The machine is top heavy. Do not tip.

Pallet Jack or Fork Lift

1. Place forks under the machine.
2. Make sure forks touch both sides or front and back of the machine frame so that they do not damage internal equipment.
3. Centrally locate load on the lift forks. Keep in mind that the center of gravity of the machine is shifted towards the side of the machine with the motion platform on it.

During transportation, the axes should be prevented from moving during transport. Proper bracing should be placed between the Z-axis and pallet area to keep the axes from shaking or colliding.

Note: When transporting machine via the wheels, ensure that the leveling feet are completely lowered. Failure to do so may result in damage to the leveling feet.

Note: For questions regarding transport, please contact the factory at (773) 282-6495.

OPERATION

Power-On Procedure

1. Verify E-Stop button is pressed on the handheld pendant.
2. Verify that all cable connections listed in the Installation section have been made.
3. Plug in the machine power cord to a 120 VAC outlet.
4. Supply air and adjust supply regulator to 100psi.
5. Adjust counter balance pressure so that the x-y stage is suspended/floating. See Machine Regulators in the Hardware Introduction for more information.
6. Turn on machine power by flipping switch on back panel.
7. Reset the E-Stop button by twisting clockwise.
8. Start up the computer.
9. If the machine is equipped with a chiller or additional auxiliary devices, make sure that these are also powered on and functioning properly.

Power-Down Procedure

1. Press the E-Stop button on the handheld pendant.
2. Save any important offsets or open programs.
3. Close the Microlution mMT Control Center software.
4. All axes should now be open loop. Move the y-axis until it rests at the bottom.
5. Shut down the computer.
6. Once the computer has powered down, flip the switches on the back panel to turn off power to both the machine and computer.
7. Slowly reduce the supply air to the machine ensuring the y-axis is still resting at the bottom of the machine until the supply air pressure is zero.

Software Startup and Homing the Motors

1. Open the 363-S CNC Software by clicking the icon on the desktop.
2. Open the Setup Panel by pressing the *Setup* menu button in the upper left hand (Refer to Software Introduction section for a definition of panels).
3. Homing for each motor is part of the Phasing Setup on the left side, Figure 7.1.
4. Press the *Begin Phasing* button, illustrated in Figure 7.1, to home the X-, Y-, and Z-stage motors. The X-stage homes first. The Y-stage homes after the X-stage, and finally the Z-stage homes. After the Z-stage has homed, verify that the phasing status is complete in the phasing status window. The machine is now ready to operate.
5. If initialization fails, press the *Reset* button and then *Begin Phasing* to reinitialize.

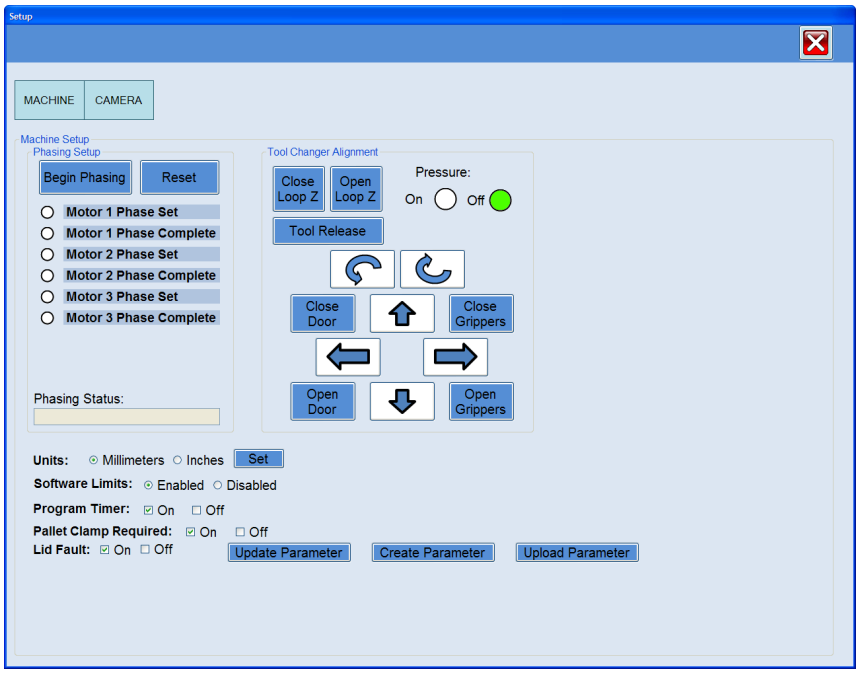


Figure 7.1 Location of the Phasing Setup controls

Spindle Operation

When running the spindle in automatic mode (refer to Spindle Control Unit sub-section) the maximum speed the software will allow is 40 000 RPM; however, this is not necessarily a safe speed for every spindle and tool.

> **Warning:** Each spindle has different requirements and limits. The user should be familiar with the specifications of the spindle and tools before use.

Before Enabling the Spindle

1. Verify that the air supply is connected to the spindle. If the spindle is an air-turbine spindle then there should be one air hose connected to the spindle. An electric spindle with an air bearing will have two air hoses. One of the air hoses will be for the air bearing and the other is for cooling. An electric spindle with ceramic bearings will have one air hose connected to it for cooling air. Check with the spindle user's guide to ensure all connections are properly setup.
2. Verify that the electrical connection between the spindle and the 363-S is secure.

Starting the Spindle

1. From the CNC Control Panel, the spindle is controlled from the Program Tab.
2. Click the *Enable Spindle* button to enable the air supply to the air bearings (if equipped).
3. Click the *Start* button to enable the air supply to the turbine and set the target speed in the RPM text box (see Figure 7.4).

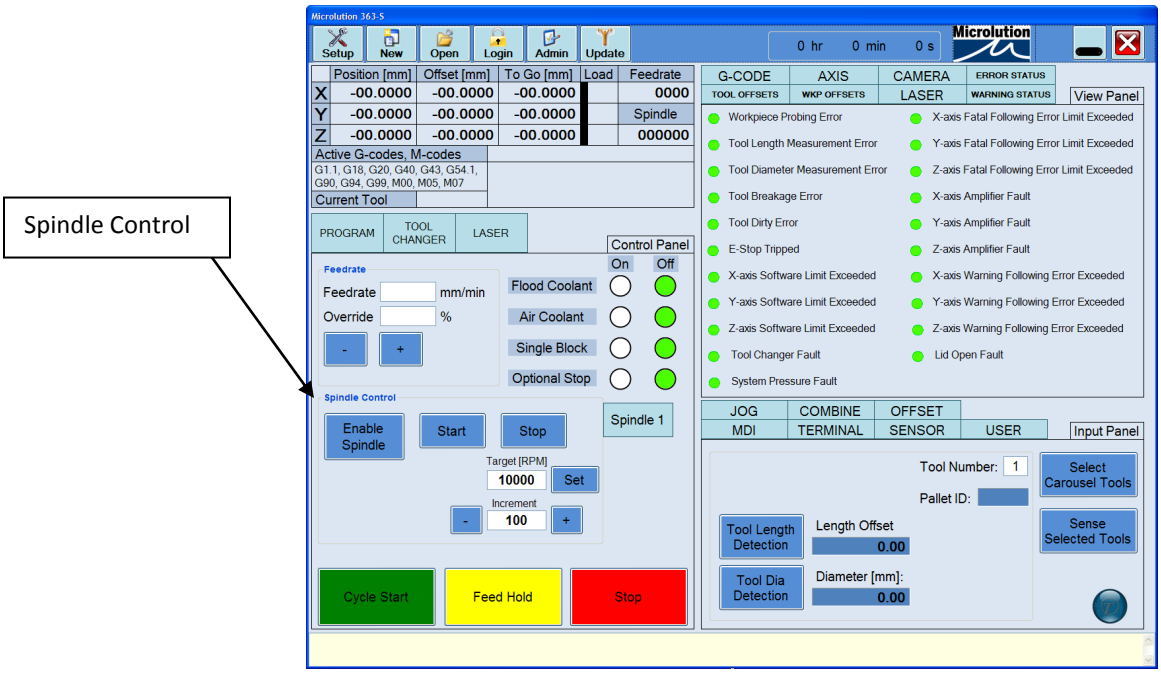


Figure 7.4: Program Run Tab of the Control Panel

The following instructions pertain to Figure 7.4.

Changing the Spindle Speed

1. Entering a new RPM value into the text box and clicking the *Set* button will change the spindle speed to the new value.
2. Using the *+/-* buttons will increase or decrease the speed of the spindle by the amount shown in the textbox.

Disabling the Spindle

1. Click the *Stop* button to stop the spindle from rotating.
2. Wait for the spindle to come to a complete stop.
3. Click the *Disable Spindle* button.
4. The spindle is now disabled and tools, pallets, or spindles can be changed by following the guidelines outlined in the Tool Changing, Pallet Changing, and Spindle Changing sections.

Tool Changing – Manual Collet – 80 000 RPM Electric/Air Bearing

This section describes how to install and remove tools from the 80 000 RPM electrically driven spindle with air/ceramic bearings and the manual collet (Figure 7.5).

Equipment

2 – M10 Wrenches

Tool Installation

1. Jog the Z stage away from pallet.
2. Turn off the spindle.
3. Verify that the spindle is clear of the workpiece and the pallet. Also verify the spindle is off.
4. Open the protective lid.
5. Place the tool into the collet with the working end towards the workpiece/pallet mounting area.
   a. If the tool has a taper, place only the widest part of the tool into the collet.
   b. Only place the shank of the tool into collet, never the flutes.
6. Place the other M10 wrench on the collet nut and turn counterclockwise until tight.
7. Remove both wrenches.
8. Close protective lid.
9. Place one M10 wrench on the lock nut close to the spindle.
10. Standing in front of the machine, turn the collet nut counterclockwise with fingers until finger tight.

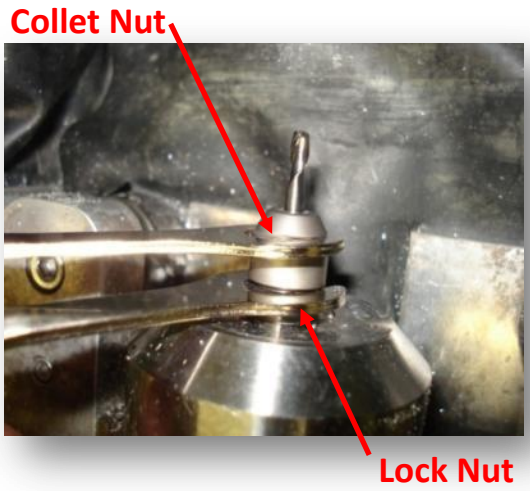


Figure 7.5: 80 000 RPM spindle tool changing

Tool Removal

1. Jog Z stage away from pallet.
2. Turn off spindle.
3. Verify tool is clear of part and pallet and spindle is off.
4. Open protective lid.
5. Place one M10 wrench on the lock nut close to the spindle.
6. Place the other M10 wrench on the collet nut and turn clockwise until loose.
7. Remove wrench from collet nut.
8. Loosen collet nut further with fingers.
9. Remove wrench from lock nut.
10. Remove tool from collet.
11. Close protective lid.

Tool Changing–Manual Collet–160 000 RPM Air Turbine/Air Bearing

This section describes how to install and remove tools from the 160 000 RPM air turbine and air bearing spindle with a manual collet (Figure 7.6).

Equipment

1 – 2.5mm hex wrench
1 – Collet pin
1 –M5.1 Wrench

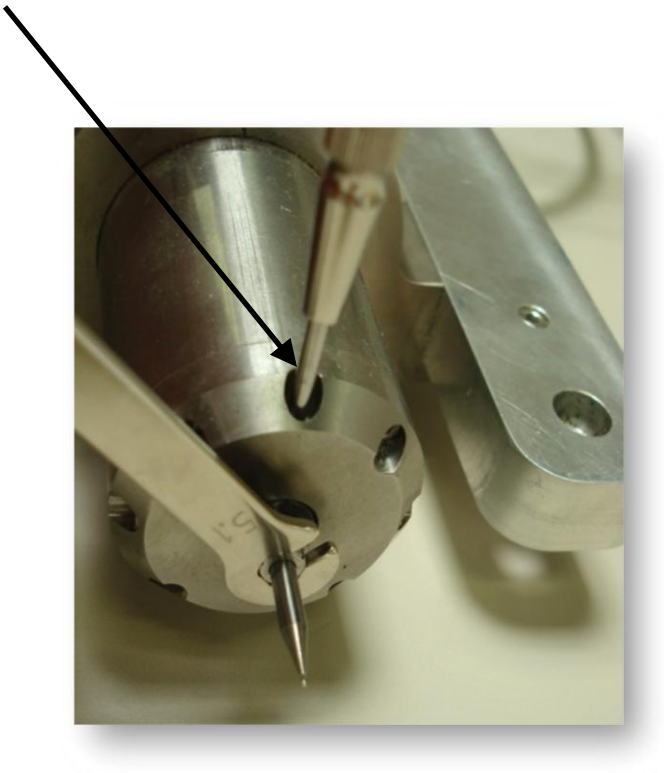


Figure 7.6: 160 000 RPM spindle tool changing

Tool Installation (Figure 7.6)

1. Jog the Z stage at least 40mm away from pallet.
2. Turn off the spindle.
3. Verify that the spindle is clear of the workpiece and the pallet. Also, verify that the spindle is disabled within the CNC control Panel.
4. Open the protective lid.
5. Place the tool into the collet with the working end towards the workpiece/pallet mounting area.
   a. If the tool has a taper, place only the widest part of the tool into the collet.
   b. Only place the round part of tool into the collet, never the flutes.
6. Remove the hex screw from the spindle with 3/32” hex wrench.
7. Place the collet pin in the hole where the hex screw was removed.
8. Standing in front of the machine, turn the collet nut counterclockwise with fingers until finger tight.
9. Place the M5.1 wrench on the collet and turn counterclockwise until tight.
10. Remove the wrench and collet pin.
11. Replace the hex screw in the spindle with a 3/32” hex wrench.
12. Close the protective lid.

Tool Removal (Figure 7.6)

1. Jog the Z stage at least 40mm away from the pallet.
2. Turn off the spindle.
3. Verify that the tool is clear of the workpiece and pallet. Also, verify that the spindle is off.
4. Open the protective lid.
5. Remove the hex screw with 3/32” hex wrench.
6. Place the collet pin in the hole where the hex screw was removed.
7. Place M5.1 wrench on the collet.
8. Standing in front of the machine, turn clockwise until loose.
9. Remove the wrench from the collet.
10. Loosen the collet nut further with fingers.
11. Remove the tool from the collet.
12. Remove the collet pin from the spindle.
13. Close the protective lid.

Tool Changing – Automatic Collet

This section describes how to install and remove tools from spindles equipped with an automatic collet (Figure 7.7).

Figure 7.7: Automatic tool changing collet

Tool Installation

1. Jog the Z stage away from the pallet.
2. Ensure that the spindle is disabled in the Input Panel, Jog Tab.
3. Verify that the spindle is clear of the workpiece and pallet. Also, verify that the spindle is off.

Note: Tool will be expelled from collet by air pressure when tool is released. Hold the tool in place when releasing or clamping the tool to prevent damage to the tool.

4. Release the collet using the *Release Tool* button on the Jog Tab of the Input Panel (Figure 7.8)

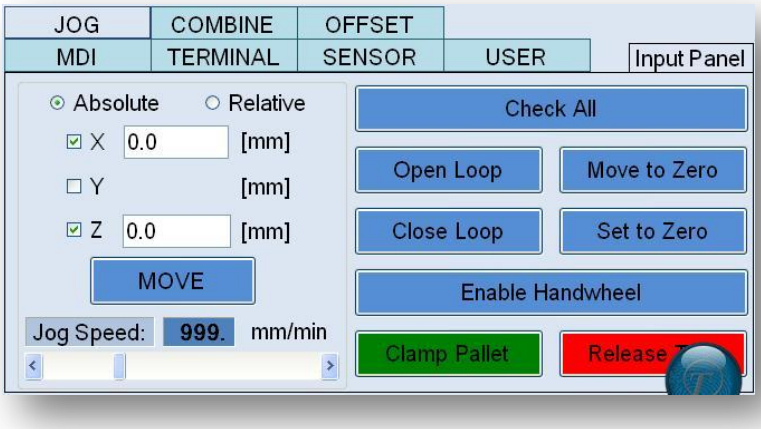


Figure 7.8: Jog Tab

5. Open the protective lid.
6. Place the tool into the collet with the working end towards the workpiece/pallet mounting area.

a. If the tool has a taper place only the widest part of the tool into the collet.
b. Only place the round part of the tool into the collet, never the flutes.
c. The automatic collet will pull the tool slightly into the spindle when clamped.

7. Clamp the tool using the *Clamp Tool* button located in the Program Run Tab of the Control Panel. The *Clamp Tool* button is in the same location as the *Release Tool* button.
8. Verify that the tool is clamped by pulling lightly on the tool.

Warning: Tools are sharp. Do not touch the cutting surfaces of the tool. Only handle tools by the taper or the shank.

9. Close the protective lid.

Tool Removal

1. Jog the Z stage away from the pallet.
2. Turn off the spindle.
3. Verify that the tool is clear of the workpiece and pallet. Also, verify that the spindle is off.
4. Release the collet using the *Release Tool* button on the Program Run Tab of the Control Panel (Figure 7.8).
5. Open the protective lid.
6. Remove the tool from the collet.

Warning: Tools are sharp. Do not touch the cutting surfaces of the tool. Only handle the tool by the taper or the main body.

7. Close the protective lid.

Tool Changing - Automatic Tool Changer

The automatic tool changer automatically loads and unloads tools from the machine. Figure 7.9 shows a layout of the hardware components.

Table 7.1: Tool requirements for the automatic tool changer

Description	Specification
Tool Length (L)	38.1 mm (1.5”)
Tool Shank Diameter (D_s)	3.18 mm (0.125”)
Tool Collar Outer Diameter (D_c)	9mm Diameter (standard size)
Tool Collar Thickness (t)	7mm (standard size)
Tool Collar Location (A)	19mm (0.75”) from tool tip

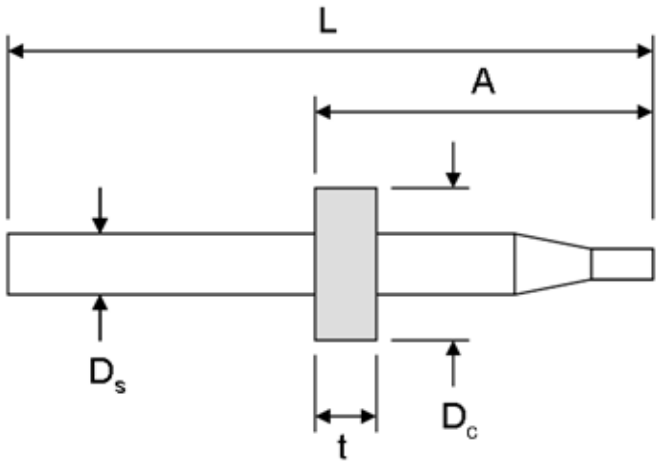


Figure 7.9: Tool dimensions

Manual Operation Mode

The touchscreen interface enables manual operation of key tool changer functions. These operations can be found in the Tool Changer Tab (shown in Fig. 7.10). Also, Table 7.2 gives a description of the manual interface functions.

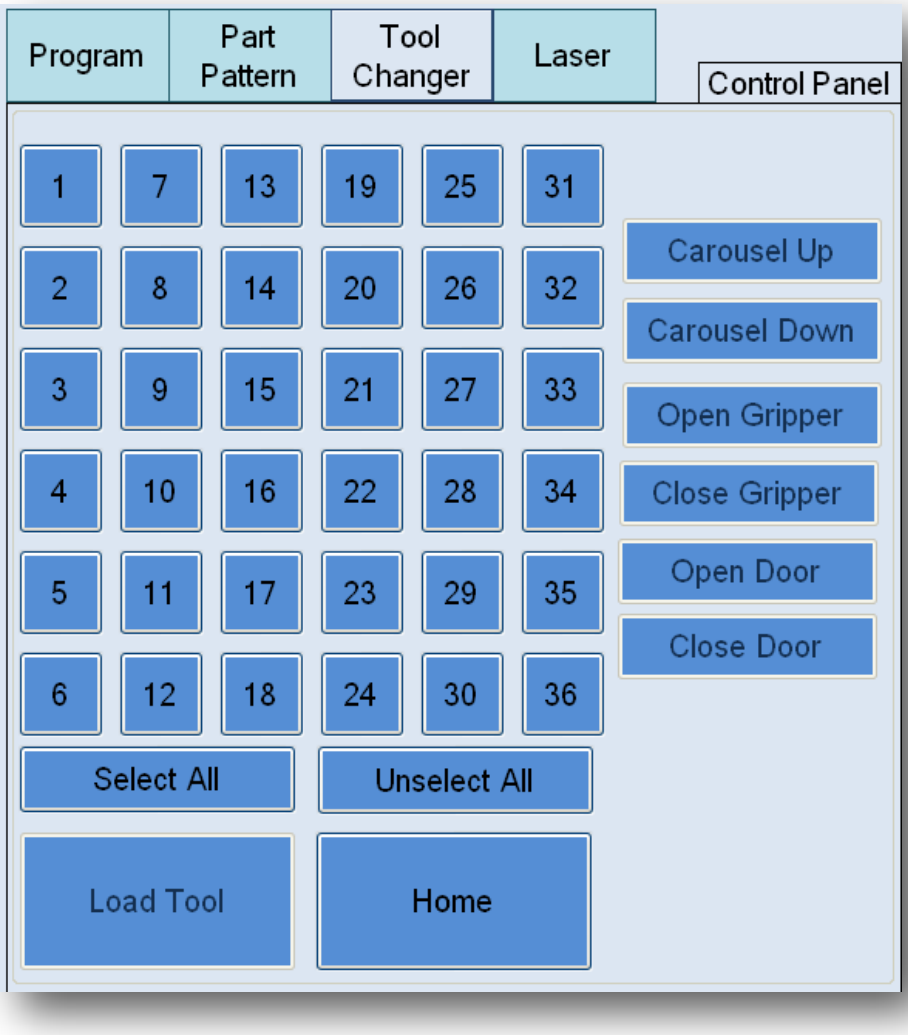


Figure 7.10: Manual interface for tool changer operation

Table 7.2: Tool changer manual interface buttons and their functions

Button	**Function**
Home	Resets communication between the PC and the tool changer controller. In addition, it performs a homing move with the tool carousel to re-establish the proper carousel orientation.
Carousel Up / Down	Raises and lowers the carousel arm. Ensure that the carousel is raised before executing a tool change. Lower the carousel arm to add/remove individual tools or add/remove entire carousels.
Open / Close Gripper	Manually opens or closes the tool grippers.

Open / Close Door	Manually opens or closes the door separating the milling area from the tool changer compartment. Ensure that the door is closed before executing machining sequences.
Selection of Tools / (Un)Select All	These buttons allow an operator to keep a visual record of tools loaded in the carousel as shown in Figure 7.11.
Load Tool	This button disables all tool buttons that are not selected and allows the operator to select which active tool to load.

Note: When loading a tool, the machine will first attempt to unload any tool in the spindle. This tool will be returned to the carousel position from which the last tool was loaded.

Warning: When machine power is cycled, the software will not retain the current spindle tool number and the carousel will be homed. Ensure correct tool carousel alignment with the Index Carousel button to prevent tool changer errors during spindle tool placement in the carousel.

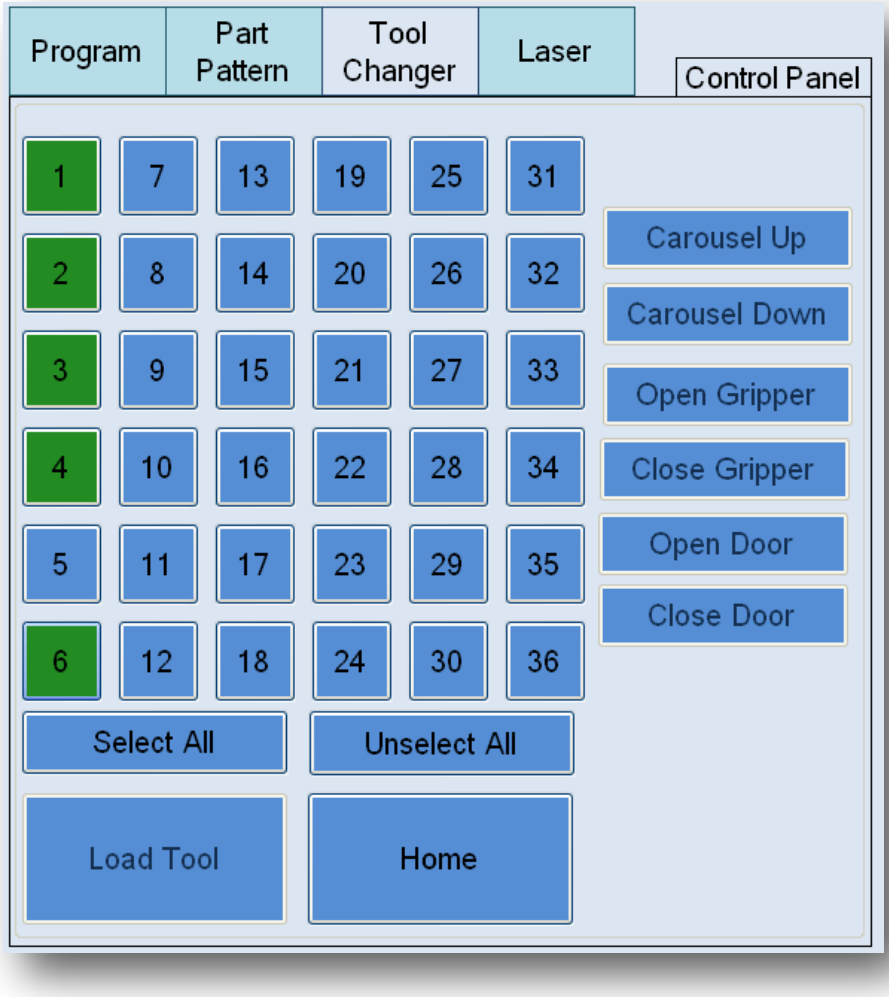


Figure 7.11: Selection of active tools – active tools shown are highlighted in green.

Operation in an NC program

M6 executes a tool change operation. The syntax for this command is as follows:

Txx M6

where xx is the tool number to be loaded. Standard implementation within an NC program might look as follows (below is a description of each step):

…
T2 M6
S10000 M3
….

1. T2 M6 (Tool Change). This command will unload the current tool in the machine, return it to the tool carousel, and load the specified tool in the spindle (in this case, Tool 2 will be loaded). In addition, this command has built in functionality to stop spindle rotation before unloading the current tool, so an M5 command preceding this command is optional but not required.
2. S10000 M3 (Start spindle rotation). This command will start spindle rotation (in the clockwise direction) at the specified speed. In this case the spindle speed is set at 10,000 rpm.

Tool Changer Alignment

The automatic tool changer can be re-aligned if tool placement into the tool wheel or spindle begins to function with error. The position of the tool gripper can be adjusted in both vertical and horizontal directions using the end stops on the main X axis and rotary pneumatic actuators. Fig 7.12 shows the horizontal X and vertical Y adjustment locations.

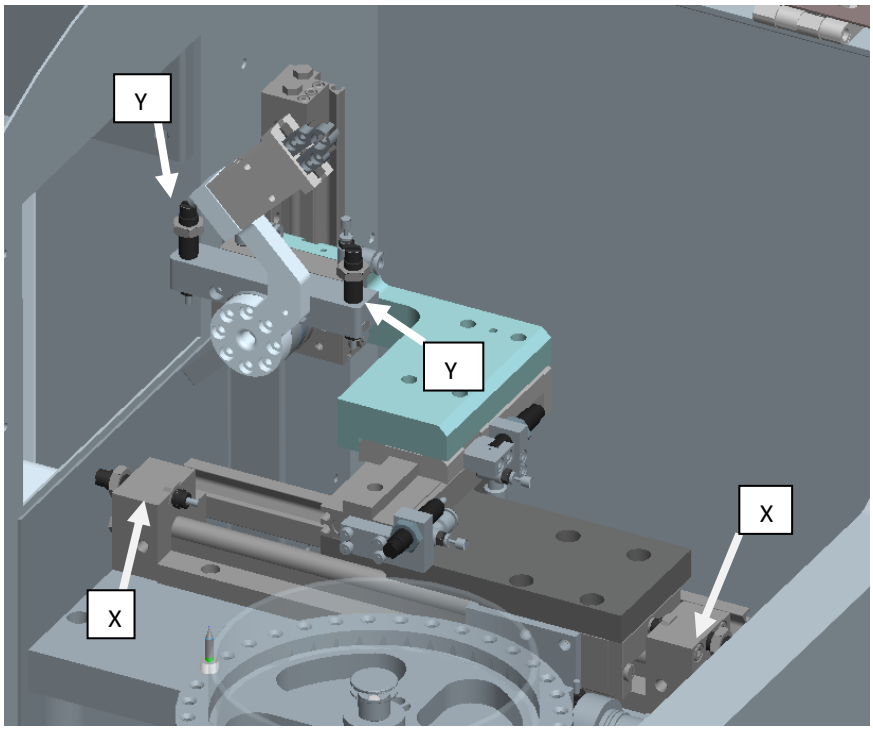


Figure 7.12: Tool changer position adjustment locations.

Before adjusting the position of the tool changer gripper arm, it is important to first run a home sequence from the tool changer software tab. Also, be sure to open the tool changer door and leave it open until adjustments are complete. The top rear panel of the machine may need to be removed to gain better access as well.

When adjusting the position of the gripper on the tool wheel side of travel, use the outer side X actuator travel stop and the top side rotary actuator travel stops for horizontal and vertical adjustment respectively. Note, vertical adjustments may affect horizontal position slightly. Fig 7.13 shows the adjustment locations and corresponding gripper travel for aligning at the tool wheel side.

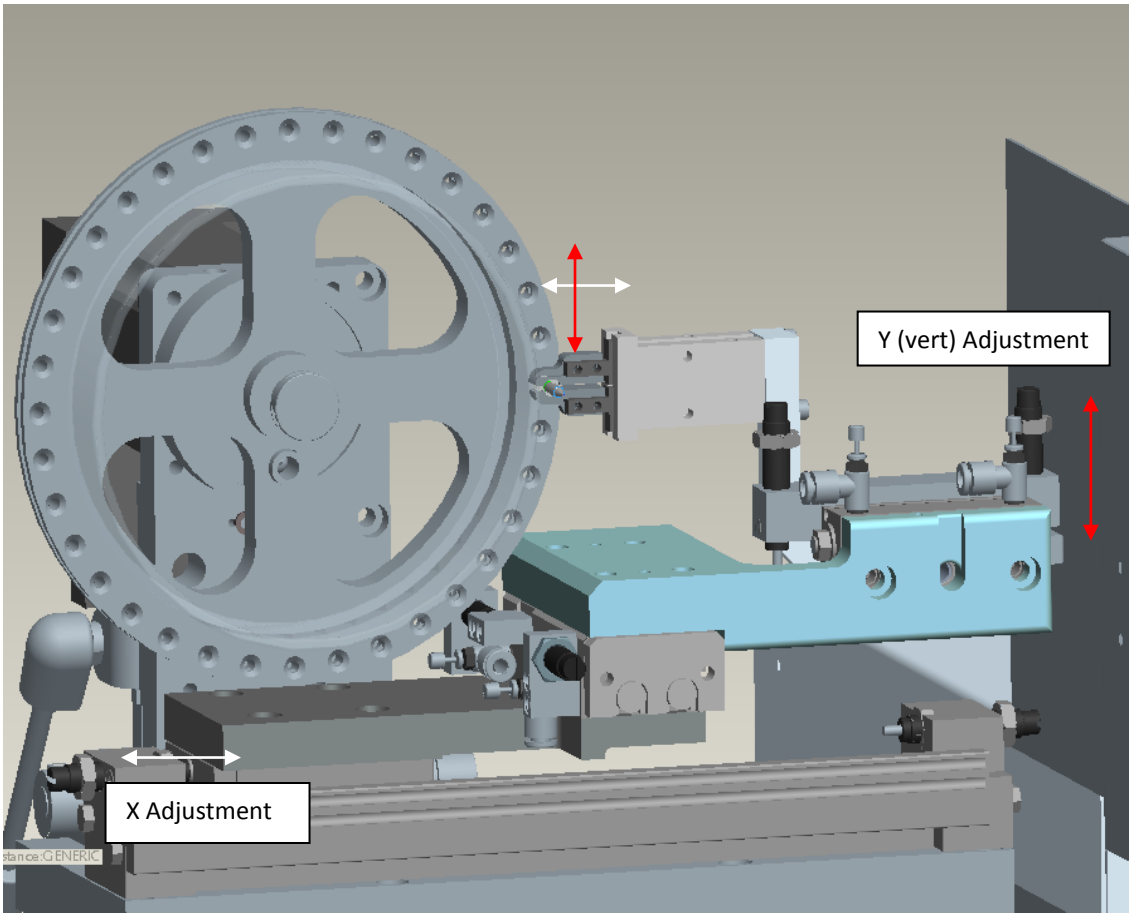


Figure 7.13: Alignment at the tool wheel side.

With the tool wheel homed, adjust the gripper so that the closed gripper is centered on the nearest tool position on the wheel.

To adjust the tool changer gripper on the spindle side, extend the tool changer arm to the spindle position and close the gripper around a tool. Make sure the spindle collet is open and empty and Kill and retract the Z stage. Adjust the vertical and horizontal position of the tool gripper so that the tool slides smoothly in and out of the open collet. Fig 7.14 shows the adjustment of the tool changer gripper on the spindle side.

Tool Collar Setting using Bohan Tool Setter

There are two means to ensure reliable tool collar setting. Many tool vendors will offer tools with pre-set tool collars. Also, a specialized tool collar setting machine can be acquired for more accurate tool collar setting. Below describes one option of tool collar setter along with the procedure for using it. This tool collar setter was purchased from Bohan Engineering Limited.

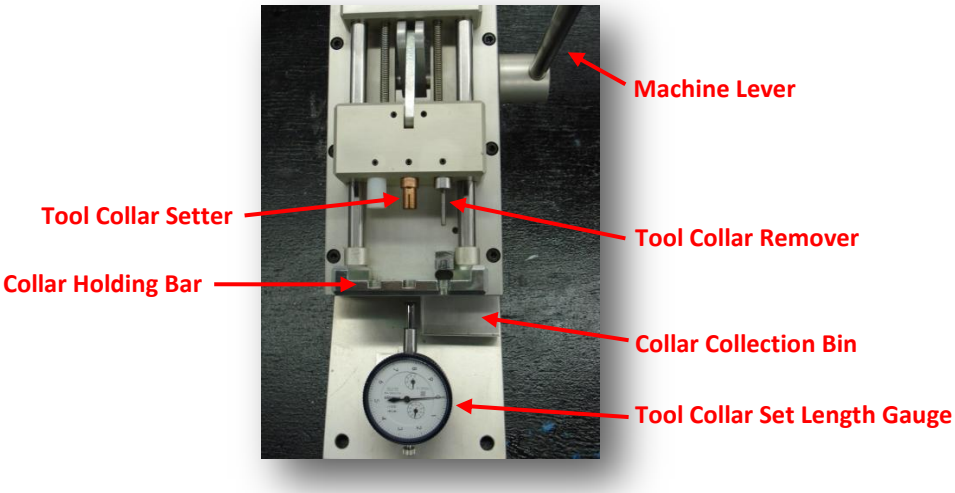


Figure 7.14 Tool Collar Setting Machine

Tool Collar Setting

Insert a tool, tip down, into the tool collar setter and a collar into the collar holding bar, as shown in figure 7.15. Rotate the machine lever downward until tool collar set length gauge is at the appropriate reading. Slowly release the machine lever and remove tool with collar from the tool collar setter.

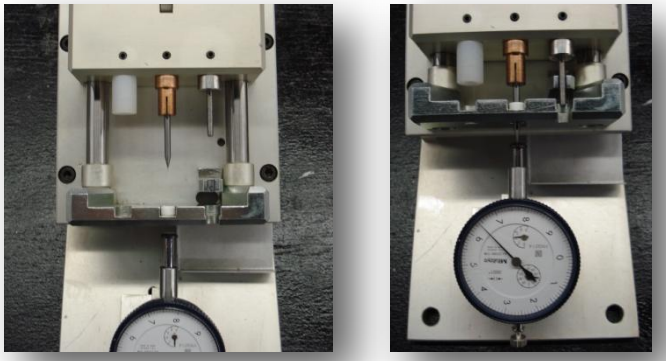


Figure 7.15 Tool Collar Setting

Tool Collar Removal

Insert tool with collar tip down with the collar resting in the collar holding bar. Rotate the machine lever downward. Make sure to hold the tool so that it does not fall from the machine and break. Once tool is free from collar slowly release the machine lever. The collar will fall into the collar collection bin for easy retrieval.

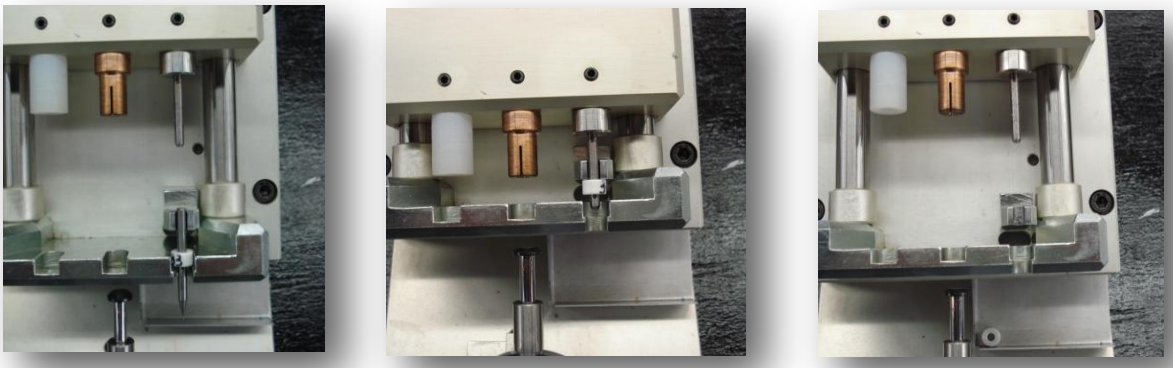
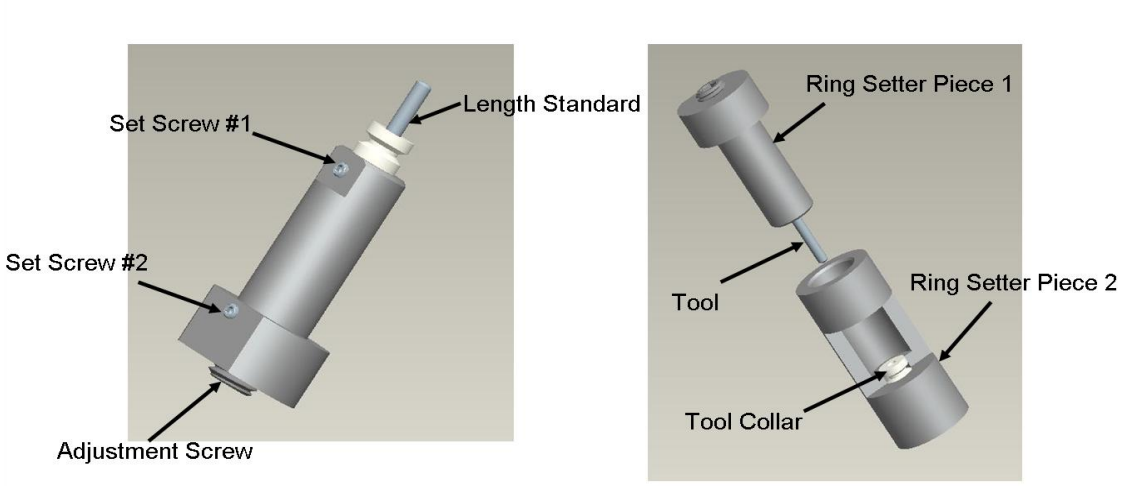


Figure 7.16 Tool Collar Removal

Tool Collar Setting using Two-Piece Setting System

1) Set desired set length
    a. Loosen set screws #1 and #2
    b. Slide one of the included length standards into part 1 of the ring setter. Back out the adjustment screw far enough to allow the collar of the length standard to rest against the ring setter.
    c. Tighten the set screw #1 to hold the standard tightly
    d. Rotate the adjustment screw until it comes to rest against the length standard. Tighten set screw #1 to fix the adjustment screw's position
2) Pressing on a tool collar
    a. Slide a tool shank first into Part 1 of the ring setter until it is touching the adjustment screw. Tighten set screw #1 to hold the tool in place.
    b. Slide a tool collar into the pocket in Part 2 of the ring setter.
    c. Press the two halves of the ring setter together. Pull apart the two halves.
    d. Loosen set screw #1 to release the tool.
3) Removing a tool collar
    a. Slide the removal pin into Part 1 of the ring setter. Tighten set screw #1 to hold the pin in place.
    b. Slide the tool into Part 2 of the ring setter.
    c. Press the two halves of the ring setter together. Pull apart the two halves.



Tool Sensing – G120 Cycle

The tool sensing function uses a Renishaw NC-4 non-contact tool setter measure tool length and diameter. The tool sensor is attached to the machine at the factory. Do not attempt to remove. Consult factory if modification is necessary.

Syntax and Parameters

G120 B.. H.. [S.. X.. Z.. J..]

B: Measurement Type (required)
- 1 = Length only
- 2 = Length and single-sided diameter measurement
- 3 = Length and double-sided diameter measurement

H: Tool Length Register (required - use tool number of tool to be measured)

S: Spindle Speed (optional – defaults to 5000, use 0 for no rotation)

X: Radial Offset for Length Measurement (optional – defaults to 0.0mm)

Z: Axial Offset for Diameter Measurement (optional – defaults to 0.5mm)

J: Additional Length Offset (optional – defaults to 0.0mm)

Length Measurement Examples:

Example 1)

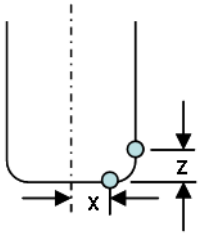
T1M6	- Load tool 1
G120 B1 H1	- Length only measurement for tool 1, measured at 5,000rpm

Example 2)

T2M6	- Load tool 2
G120 B3 H2 S30000	- Length and diameter measurement for tool 2, measured at 30,000rpm

Calculation of X and Z parameters:

X and Z are used to change the measurement position for the laser measurement cycle. The X parameter should be used when measuring the length of tools larger than 1.5mm in diameter, and the Z parameter should be used when the standard axial position (0.5mm from the tip) will not measure the correct diameter, such as for very small diameter endmills or endmills with corner radii. The figure below shows how to calculate X and Z for larger tools. For small tools where Z=0.5 is too large, use Z = (length of cut)/2 or Z=0.1, whichever is larger.



X = D/2 – corner rad. – 0.5mm
Z = corner rad. + 0.5mm

Auto-Sensing Multiple Tools for a Job using the Microlution mMT Control Software

1) In the Tool Changer Tab, Press SELECT TOOLS.
2) Select the individual tools that will be used for the job (tools should highlight green when selected)
3) Press SELECT TOOLS again to finish selection
4) In the Sensor Tab, Press SENSE SELECTED TOOLS. A program will automatically be written to the MDI and executed.

Manual Operation Mode

Below are the steps to use the workpiece sensor manually using the touch screen interface.

1. Make sure a tool is loaded in the spindle.
2. Press tool length or the tool diameter sensing button in the Sensor Tab.
3. The measurements are saved to data registers for tool number 1.
4. The measurement data also appears in the text box next to the button.

Implementation in an NC Program

Standard implementation within an NC program might look like this (below is a description of each step):

```
…
T1 M6
G120 B1 H1
G54
G43 H1
…
```

1. T1 M6 (Load tool number 1).
2. G120 B1 J0 H1 (Tool measurement cycle). This command will measure length and diameter for tool number 1 using the length experience (J) parameter.
3. G54 (apply workpiece offset). This command is required every time prior to applying a tool length offset.
4. G43 H1 (apply tool length offset). This command applies the tool length offset for tool 1.

Z-Axis Fixture Probing – G123 Cycle

The G123 cycle performs a z-axis probing cycle using force-feedback from the machine's linear motors to determine the trigger location. This is feature is only available for select spindle options. The cycle will first perform a linear move to the measurement start location (if X, Y, or Z passed). Next, the machine moves slowly in the –Z direction until a trigger is detected. The machine moves away from the fixture a small amount and repeats the triggering move. Once the trigger is detected again, the Z-axis moves 10mm away from the triggering location and the work offset registers are updated.

Syntax and Parameters

G123 W.. H.. [X.. Y.. Z.. J..]

W: Work offset to update (required)

0=current offset, 1=G54, 2=G55, 3=G56, 4=G57, 5=G58, 6=G59

H: Tool Length Register (required - use tool number of active tool)

X: X-axis start location for measurement (optional – defaults to current location)

Y: Y-axis start location for measurement (optional – defaults to current location)

Z: Z-axis start location for measurement (optional – defaults to current location)

WARNING

CHANCE FOR COLLISION IF IMPROPER VALUE PASSED – USE ONLY IF YOU KNOW THE APPROXIMATE WORK OFFSET

J: Additional Z-axis Offset (optional – defaults to 0.0mm)

Fixture Probing Example

T1M6	- Load tool 1 (tool must have been measured previously using G120)
G54 G43 H1	- Apply work and tool offsets (use if approximate offset is known)
G123 W1 H1	- Measurement cycle, updates the Z-axis register of G54

Workpiece Probing

This section of the manual describes the functioning of the probing routines on the 363-S using a probe attached to the spindle collet.

Probe guidelines

A probe is a precision tool with which a measurement is performed. The probe is clamped in the spindle collet and just like a machining tool. The probe needs to have an accurate and end zone which is concentric to the tool shank in the form of a small sphere or cylinder which would be used for the actual measurement. Recommended diameter for the end zone is 2mm which would enable measurements of smaller features. An example of a probe would be a tool blank (no flutes) of 2mm diameter at the end which is ground well on a 1/8" shank size.

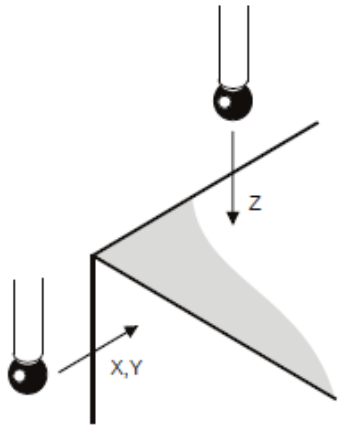
Triggering

The trigger is set up to detect any difference between the commanded position and the actual position during measurements. When the difference exceeds a preset value, a reading is taken and stored as the measured value. To create this difference, the motors apply a limited amount of force in order to "feel" when the probe makes contact with the measurement surface. However, care should taken not to measure any thin wall or delicate features as the force may be enough to cause permanent damage to the part. Follow the same procedures as during a cutting process to keep hands out of the machine to prevent possibility for operator injury.

Precautions

Before performing the measurements ensure that the correct work offset and tool offsets are applied. The g-codes are designed to operate in the currently active offset although they can update any other work or tool offset. Also, ensure that the correct measuring probe is loaded into the spindle. Measuring with a fluted tool is not recommended because the probing force can damage the work piece and/or the tool. If the measuring program fails to execute for some reason then an error is outputted which may help debug the error source and the program stops immediately. Please return the stages to the start position before re-running the code.

X Y Z single surface measurement (G65 P9811)



Description

The cycle measures a surface to establish the size or position.

Application

With its tool offset active, position the probe adjacent to the surface. The cycle measures the surface and returns to the start position. The measured surface can be considered in one of two ways:

1. As a size, where the tool offset is updated in conjunction with the Tt input.
2. As a reference surface position, for the purpose of adjusting a work offset using the Ss input.

The function supports single axis measurement moves only. The appropriate probe length and diameter must be stored in tool number which is passed as parameter H. This can be done by using the tool sensor and running a diameter and length measurement cycle.

Format

G65 P9811 Xx or Yy or Zz Hh[Cc Ee Ff Qq Ss Tt Uu Vv]
Where [] denotes optional inputs.

Compulsory inputs

Xx	x=	
or		
Yy	y=	The surface position or size in mm.
or		
Zz	z=	
and		
Hh	h=	Tool offset number storing the appropriate diameter for the measuring probe. e.g. H1 will use the tool radius value stored for tool 1.

Optional inputs

Cc	c=	Measurement mode Sets the measurement mode for the machine. Value can be either 0 (regular mode) or 1 (fine mode). Default value – 0 (regular mode). e.g. C1 will set the machine to fine measuring mode.
Ee	e=	Experience Value Specify an experience factor value to be applied to the measurement. Value can be either positive or negative. Default value – 0.

e.g. E0.1 causes the measurement to be shifted by 0.1mm

Ff f= Feedback Percentage

The percentage feedback that is used when updating an offset. Enter a value between 0 and 2 (0% and 200%). Default value – 1.

Qq q= Overtravel distance

The probe overtravel distance for use when the default values are unsuitable. The probe will then travel beyond the expected position when it searches for a surface. Default value – 2mm in fine mode and 3mm in regular mode.

Ss s= The work offset number that will be set

S1 to S6 (G54 to G59)

S101 to S494 (G54.1 P1 to G54.1 P394)

New work offset = Existing offset value + error

Tt t= The tool offset number to be updated

Updates the tool radius comp for X and Y measurements and the tool length comp for Z measurement.

T1 to T36 (Tool 1 to Tool 36)

New tool offset = Existing offset value + error

Uu u= Upper tolerance limit

If this value is exceeded no tool offset or work offset is updated and the cycle stops with an alarm. Where applicable, this tolerance applies to both the size and position. Default value – 5mm.

e.g. U1 sets the upper tolerance limit to 1mm.

Vv v= Null band

This is the tolerance zone in which no tool offset adjustment occurs. Default value – 1micron.

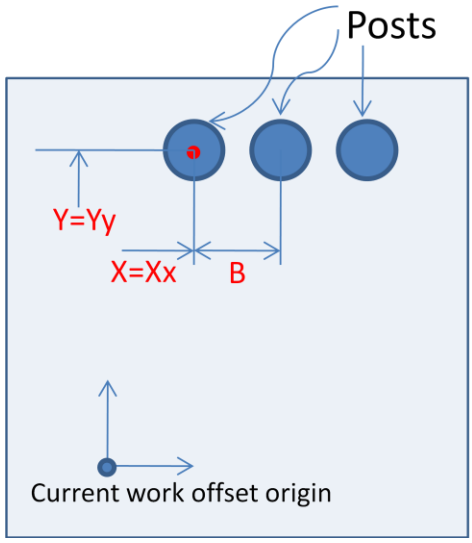
Example: Measuring a single surface in X

T1 M6	Load appropriate measuring probe
G54 X0 Y0	Apply correct work offset and go to start position
G43 H1	Activate tool length offset for particular tool
G1 X-20 Y0	Goto appropriate X,Y location for measurement
G1 Z5	Goto appropriate Z location for measurement
G65 P9811 X-10 Q5 S2 U1 V0.01	Call single surface measurement

continue with program…

The X offset for G55 (S2) will be updated by the error of the surface position.

Pallet ID function (G65 P9809)



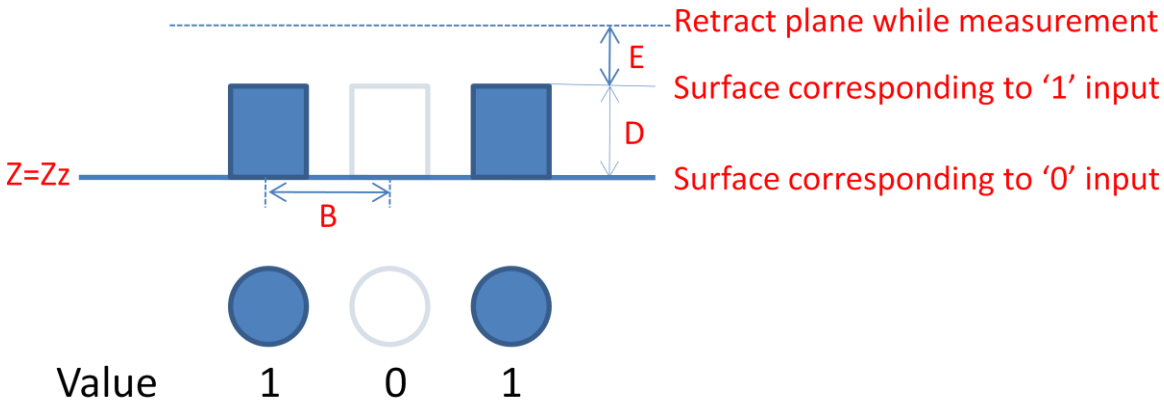
A: No. of posts
B: Spacing of posts along X
D: Height of the posts
E: Clearance above the posts

Description

The cycle measures a specified zone on the pallet to identify the pallet number.

Pallet identification geometry description

The pallet identification function identifies the 'posts' as features of different Z heights. The shape of the post is not as important (can be a circle, square or any other shape) as the height difference between a post corresponding to '0' input and a post corresponding to '1' input. The figure below illustrates the various parameters associated with the posts.



Pallet orientation

The pallet identification (ID) function also works as a pallet orientation function. If the pallet is attached rotated on the machine the pallet identification function will not execute completely if the surface in the measuring zone is at a different height than expected by the function. On the other hand, a surface at the same height as the Z parameter for the pallet identification will not terminate the function and might output a false pallet id. Hence, to use the function as a pallet orientation function as well it is necessary that the measuring posts be at a height different from the height of the rest of the surface. A tighter control for the measuring band can be provided with the help of parameter Ee (as explained below) which governs the clearance height from where the measurement begins. This helps reduce the possibility of a false trigger and hence a false pallet ID output. As a better practice, it is advised not to use that least and the highest value numbers as valid pallet IDs, e.g. For a 3 post measurement, consider 0-0-0 and 1-1-1 as invalid pallet IDs.

Application

The function measures posts at specified locations to identify the pallet number for the current pallet. Based on the output, appropriate machining or measurement code can be used on the machine.

The measurement assumes that the posts are located along an horizontal line beginning at point (X,Y) and subsequent posts are at a spacing of Bb each along the positive X direction. The Z height of the post also needs to be specified as a compulsory input. The program runs in the currently active work offset.

Measurement system

The measurement is done in the binary system. For measurement with more than 1 post, the left most post represents the most significant bit (MSB) and the right most post representing the least significant bit (LSB). The weightage of each post from the right is obtained by multiplying the weightage of the post to the right of it by 2. By definition, the weightage of the right most post is 1. Hence, by definition, the weightage of the post to the left of it is 2 and one left to that is 4 and so on. Consider the example below with 3 posts,

Value of the posts	1	0	1
Weightage of the posts	4	2	1

Output in binary system - 101

To obtain the output in the decimal system, multiply the value of the post by its individual weightage.

Hence the output for this case, in decimal system, is 1*4 + 0*2 + 1*1 = 5

The program outputs the above pallet number in the decimal system in variable Q4325.

Format

G65 P9809 Xx and Yy and Zz [Aa Bb Cc Dd Ee]

Where [] denotes optional inputs.

Compulsory inputs

Xx x = X position of the first identification post in mm in absolute co-ordinates.

and

Yy y = Y position of the first identification post in mm in absolute co-ordinates.

and

Zz z = Z height of the surface corresponding to a '0' input value (no post present) in mm in absolute co-ordinates. The height of the post above this surface is specified by parameter Dd.

Optional inputs

Aa a = Number of posts

Specifies the number of posts used by the pallet identification code. Default value – 3. Only positive input acceptable.

Bb b= Spacing of the posts along X

Specifies the spacing of the posts along positive X direction. Default value – 5mm. Only positive input acceptable.

Cc c= Measurement mode

Sets the measurement mode for the machine. Value can be either 0 (regular mode) or 1 (fine mode). Default value – 0 (regular mode).

e.g. C1 will set the machine to fine measuring mode.

Dd d= Height of the posts

Specifies the height of the post above the Zz input parameter when a post is present. Default value – 2mm. Only positive input acceptable.

Ee e= Clearance above the posts

Specifies the clearance value for the probe to retract above the posts. Default value – 2mm. Only positive input acceptable.

Example:

T1 M6	Load appropriate measuring probe
G54 X0 Y0	Apply correct work offset and go to start position
G43 H1	Activate tool offset for particular tool
G65 P9809 X-10 Y10 Z-20 A4 B3 D3 E4	Call pallet identification function

continue with program…

Program will output the pallet number in the decimal system in variable **Q4325**.

Error code index table

If any of the probing functions fails to complete as commanded, it may be due to one of many reasons. If this were to happen the error flag Q801 will have a non-zero value corresponding to the type of error that aborted the function. The table below provides detailed description for each error number and the function in which the error appeared.

ERROR FLAG	REASON	CYCLE NUMBER
1	INSUFFICIENT PARAMETERS IN FUNCTION CALL	9811
2	INCORRECT USE OF S AND T PARAMETERS	9811
3	PROTECTED MOVE FAILED	9809
4	NO SURFACE DETECTED	9811
5	G31 FAILED TO INITIALIZE OR EXECUTE	9811
6	OFFSET UPDATE TYPE NOT ALLOWED	9811
7	OFFSET UPDATE MORE THAN UPPER LIMIT	9811
9	F INPUT OUT OF RANGE	9811
12	Q INPUT OUT OF RANGE	9811
13	S INPUT OUT OF RANGE	9811
14	H OR T INPUT OUT OF RANGE	9811
15	U INPUT OUT OF RANGE	9811
16	V INPUT OUT OF RANGE	9811
21	INSUFFICIENT PARAMETERS	9809
22	INCORRECT PARAMETERS IN FUNCTION CALL	9809
23	TRIGGER ERROR ON SURFACE	9809

Example

The example does the following operations –

a. Measure a feature at 2 points along X direction to determine the center of the feature along X,
b. Update the value of the G55 (Parameter 1 – Arbitrarily selected) X offset accordingly to center the co-ordinate system on the feature,
c. Apply G55 (Parameter 1) offset,
d. Take a Z measurement at the center,
e. Set the G55 Z offset such that Z=0 is 6mm (Parameter 2 – Arbitrarily selected) above the surface.

The example starts with the approximate work offsets stored in G54 (Parameter 3 – Arbitrarily selected) and the tool length and diameter stored in tool offset number 2 (Parameter 4 – Arbitrarily selected). At the end of the program the correct work piece offsets are stored in G55 (Parameter 1 – Arbitrarily selected).

G56 (Parameter 5 – Arbitrarily selected) and G57 (Parameter 6 – Arbitrarily selected) are used to stored temporary values during the cycle.

Code

G10 L2 P2 X(Q2000)	Copy the X offset from G54(Parameter 3) to G55(Parameter 1) L2 P2 corresponds to G55 (Parameter 1) Q2000 is the X offset for G54 (Parameter 3)
G4 P0.01	Dwell
G10 L2 P2 Y(Q2001)	Copy the Y offset from G54(Parameter 3) to G55(Parameter 1) L2 P2 corresponds to G55 (Parameter 1) Q2001 is the Y offset for G54 (Parameter 3)
G4 P0.01	Dwell
G10 L2 P2 Z(Q2002)	Copy the Z offset from G54(Parameter 3) to G55(Parameter 1) L2 P2 corresponds to G55 (Parameter 1) Q2002 is the Z offset for G54 (Parameter 3)
G4 P0.01	Dwell
G54	Apply G54 (Parameter 1) work offset
G0 X0 Y0 G90	Change to absolute mode and move to X=0 and Y=0
G43 H2	Apply appropriate tool offset, tool 2 here (Parameter 4)
G1 Z10 F1000	Move to Z=10mm
X-30	Moves to an appropriate negative X location for measurement (Location will depend on size of feature)
Z-8	Move to appropriate Z location for measurement (Location will depend on size of feature)
G65 P9811 X-26 H2 S3	Measurement move with appropriate tool offset H2 (Parameter 4) and G56 (Parameter 5 – S3) (Location will depend on size of feature)
Z10	Retract in Z
X30	Moves to an appropriate positive X location for measurement (Location will depend on size of feature)
Z-8	Move to appropriate Z location for measurement (Location will depend on size of feature)
G65 P9811 X26 H2 S4	Measurement move with appropriate tool offset H2 (Parameter 4) and G57 (Parameter 6 – S4)
Z10	Retract in Z
X0 Y0	Move to zero position
G4 P0.1	Dwell
G10 L2 P2 X(Q2000+(Q2010+Q2015)/2)	Set X offset for G55 as previous value plus error L2 P2 corresponds to G55 (Parameter 1)

```
                                Q2000 is X offset for G54 (Parameter 3)
                                Q2010 is X offset stored in G56(Parameter 5) by measurement 1
                                Q2015 is X offset stored in G57(Parameter 6) by measurement 2
    G4 P0.1                         Dwell
    G10 L2 P3 X0                Reset X offset for G56 (Parameter 5) to zero
    G4 P0.1                         Dwell
    G10 L2 P4 X0                Reset X offset for G57 (Parameter 6) to zero
    G4 P0.1                         Dwell
    G55                         Apply G55 (Parameter 1) work offset
                                This offset has the new X offset centered on the feature
    G43 H2                          Apply appropriate tool offset, tool 2 here (Parameter 4)
    X0 Y0                           Move to zero position
    Z10                         Move to appropriate location for measurement
G65 P9811 Z0 H2 S2              Measurement move with appropriate tool offset H2 (Parameter 4) and
                                G56 (Parameter 5 – S3)
                                (Location will depend on size of feature)
                                This will update the Z offset for G56 (Parameter 5)
G10 L2 P2 Z(Q2007+6)            Offset Z offset by 6mm to have the Z axis zero 6mm above the work
                                piece surface
    G55                         Apply G55 (Parameter 1) work offset
                                Has the X offset such that tool is centered along X
                                    Has Z offset such that Z axis zero position is 6mm above the
                                surface
    G43 H2                          Apply appropriate tool offset
    M30                             End program
```

Subroutines and Macro Program Calls

Subroutine and macro program calls, hereafter jointly referred to as subprograms, can be performed by the Microlution 363-S. Subprograms can be very useful in order to reduce the amount of redundant code contained in the main program. Subprograms are also unique because they enable the use of conditional statements. Conditional statements are further explained in the next section.

Setup and Organization

Subprograms can be accessed by main programs in 3 ways. First, the subprograms can be included in the main program file as shown in the subprogram example. These subprograms should be noted by starting the code with the subprogram number (i.e. O####) and ending the code with a M99 call. The four number format is required for this to work properly. Secondly, the subprograms can be stored as separate files in the same folder as the main program. These programs must be named with only the subprogram number in the four digit format and be of the same file type as the main program. Lastly, the subprograms can be stored in the communal subprogram folder [C:\Program Files\Microlution\Subprograms]. These also need to be named with only the subprogram number in the four digit format and be of the same file type as the main program. If there is no subprogram found for a subprogram call, an error will result.

Ensure that the main program also has a four digit format program number.

Subprograms numbered 1 – 8999 are available for user access. Subprograms are 9000 – 9999 are for internal access.

NOTE

It is possible to have different subprograms in all 3 places. Ensure that the correct subprogram is in correct place. The main program will first look at the bottom of its own file, then in the same folder and then finally in the communal folder.

NOTE

Subprograms are loaded into controller memory upon the *Load G-Code* button click. If any subprogram files are modified after the *Load G-Code* button is clicked, those changes will not take effect until after the Gcode is closed with the *Close G-Code* button and then re-loaded with the *Load G-Code* button.

Variable Definition

Variables and constants can be defined with variable names. These definitions are located in the C:\Program Files\Microlution\SubPrograms\Macros.txt file. Definitions use the following syntax and cannot contain spaces.

#DEFINE *MACRO_NAME* *[VARIABLE OR CONSTANT]*

Example:

```
#DEFINE     A_PARAMETER     Q101
#DEFINE     Z_RETRACT       10
```

By placing the example code into the C:\Program Files\Microlution\SubPrograms\Macros.txt file the user will now be able to write macros using the variable names A_PARAMETER and Z_RETRACT instead of variable Q101 and constant 10.

Syntax and Parameters

Subroutines

M98 P.. [L..]
P: Subprogram number (required)
L: Subprogram repeats (optional – defaults to 1)

Macros

G65 P.. [A-M, Q-Z]
P: Subprogram number (required)
A-M, Q-Z: Macro call parameters

Subprogram Parameter Variables

Parameter	Variable
A	Q101
B	Q102
C	Q103
D	Q104
E	Q105
F	Q106
G	Q107
H	Q108
I	Q109
J	Q110
K	Q111
L	Q112
M	Q113
P	Q116
Q	Q117
R	Q118
S	Q119
T	Q120
U	Q121
V	Q122
W	Q123
X	Q124
Y	Q125
Z	Q126

Subprogram Example

```
O0000           - Main program start
G90             - Set coordinate system to absolute mode
G54             - Apply workpiece offset #1

G1 F1000        - Set linear mode and feedrate
X15 Y15 Z10     - Move to (15,15,10)
M98P0002L2      - Call subroutine 0002 twice
X15 Y-15 Z10    - Move to (15,-15,10)
M65P0003X2Y3Z5  - Call macro 0003, pass parameters X, Y, Z
X-15 Y-15 Z10   - Move to (-15,-15,10)
M98P0002L4      - Call subroutine 0002 four times
X-15 Y15 Z10    - Move to (-15,15,10)
M65P0003X5Y1Z3  - Call macro 0003, pass parameters X, Y, Z
X0 Y0 Z0        - Move to (0,0,0)
M30             - End program, rewind

O0002           - Subprogram 0002 start
G91             - Set coordinate system to incremental mode
G1 F500         - Set linear mode and feedrate
X5              - Move 5 in +x-axis
X-5             - Move 5 in -x-axis
Y5              - Move 5 in +y-axis
Y-5             - Move 5 in -y-axis
Z-5             - Move 5 in -z-axis
Z5              - Move 5 in +z-axis
G90             - Set coordination system to absolute mode
M99             - End subprogram call

O0003           - Subprogram 0003 start
G91             - Set coordinate system to incremental mode
G1 F500         - Set linear mode and feedrate
Z[-q126]        - Move in z-axis by negative Z parameter value
X[q124]         - Move in x-axis by positive X parameter value
Y[q125]         - Move in y-axis by positive Y parameter value
X[-q124]        - Move in x-axis by negative X parameter value
Y[-q125]        - Move in y-axis by negative Y parameter value
Z[q126]         - Move in z-axis by positive Z parameter value
G90             - Set coordinate system to absolute mode
M99             - End subprogram call
```

Data Setting

The G10 command can be used for data setting tool offset, tool compensation, and workpiece offset values. It can also be used to set user variables. Offset/compensation setting commands (data setting group numbers 2, 20, 10, 12, 11, 13) are non-modal and so only affect the line they are on. User variable setting commands (data setting group number 50) are modal, each following line is an additional data setting command, and can only be turned off using the G11 command.

The current coordinate system mode, absolute or incremental, affects the behavior of data setting groups 2, 20, 10, 12, 11, and 13. If in absolute mode, these groups will overwrite the current offset/compensation value with the parameter value. If in incremental mode, these groups will add the parameter value to the current offset/compensation value.

Data Setting Group Descriptions

Group Number	Description	Required Parameters	Optional Parameters
2	Fixed offset data setting group. Enables updating workpiece offsets. Parameter P denotes the offset with range 1 – 6 referring to G54 – G59.	P – Workpiece offset reference parameter	X – X-axis offset value Y – Y-axis offset value Z – Z-axis offset value
20	Additional offset data setting group. Enable updating additional workpiece offsets. Parameter P denotes the offset with range 1 – 394 referring to G54.1 P1 – G54.1 P394.	P – Workpiece offset reference parameter	X – X-axis offset value Y – Y-axis offset value Z – Z-axis offset value
10	Tool length offset data setting group. Enable updating tool length offsets. Parameter P denotes the tool with range 1 – 36 referring to H1 – H36.	P – Tool reference parameter	R – Parameter value
12	Tool diameter offset data setting group. Enable updating tool diameter offsets. Parameter P denotes the tool with range 1 – 36 referring to H1 – H36.	P – Tool reference parameter	R – Parameter value
11	Tool length compensation data setting group. Enable updating tool length compensation. Parameter P denotes the tool with range 1 – 36 referring to H1 – H36.	P – Tool reference parameter	R – Parameter value
13	Tool diameter compensation data setting group. Enable updating tool diameter compensation. Parameter P denotes the tool with range 1 – 36 referring to H1 – H36.	P – Tool reference parameter	R – Parameter value
50	User variable data setting group. Enables updating user variables. Parameter P denotes the variable with range 1-1000 referring to system variables P7000-P7999.	P – Variable reference parameter	R – Parameter value

Data Setting Examples

G90 G10 L2 P1 X0 Y10 Z7 - Set workpiece offset G54 to (0,10,7)

G91 G10 L20 P11 X-5 Y1 Z2 - Update workpiece offset G54.1 P11. Add -5 to X-axis, add 1 to Y-axis, add 2 to Z-axis.

G90 G10 L10 P3 R15 - Set tool 3 length offset to 15

G90 G10 L12 P23 R-9	- Set tool 23 diameter offset to -9
G91 G10 L11 P1 R-0.01	- Increment tool 1 length compensation by -0.01
G90 G10 L13 P14 R0.15	- Set tool 14 diameter compensation to 0.15
G10 L50	- Start user variable setting
P1 R0	- Set user variable 1 (system variable P7000) to 0
P35 R24	- Set user variable 35 (system variable P7034) to 24
P952 R-5	- Set user variable 952 (system variable P7951) to -5
P804 R0.09	- Set user variable 804 (system variable P7803) to 0.09
G11	- End user variable setting

Conditional Statements

The following logic statements are supported the Microlution 363-S. The logic statements are valid in sub-routines and macro calls only and cannot be used in a main program.

1. If [{condition}] {action}
2. If [{condition}]
   Action 1
   Action 2
   …
   Endif
3. While [{condition}] {action}
4. While [{condition}] {action}
   Action 1
   Action 2
   …
   Endwhile
5. GOTO {line number}

If [{condition}] {action}

{condition} consists of one or more sets of **{expression} {comparator}**
{expression}, joined by logical operators **AND** or **OR**
{action} is a program command

With an action statement or statements following on that line, it will execute those statements provided the condition is true. If the condition is false, it will not execute those statements.

With no statement following on that line, if the condition is true, Turbo PMAC will execute all subsequent statements on following lines down to the next **ENDIF**. If the condition is false, it will skip to the **ENDIF** and continue execution there.

In a PLC program, compound conditions can be extended onto multiple program lines with subsequent **AND** and **OR** statements.

There is no limit on nesting of **IF** conditions and **WHILE** loops (other than total buffer size) in fixed motion and PLC programs.

Supported comparators
=, !=, >, <, !<, !>

Examples:

IF [P1>10] M1=1

IF [M11=0 AND M12!=0] M2=1 M3=1

IF [M11=0 AND M12!=0]
M2=1
M3=1
ENDIF

See also user variable description for list of accessible / usable variables.

While [{condition}] {action}

{condition} consists of one or more sets of **{expression} {comparator}**
{expression}, joined by logical operators **AND** or **OR**.
{action} is a program command

This statement allows repeated execution of a statement or series of statements as long as the condition is true. It is Turbo PMAC's only looping construct. It can take two forms:

(Valid in motion program only) With a statement following on the same line, it will repeatedly execute that statement as long as the condition is true. No **ENDWHILE** is used to terminate the loop.
WHILE [{condition}] {action}

(Valid in motion and PLC programs) With no statement following on the same line, it will execute statements on subsequent lines down to the next **ENDWHILE** statement.
WHILE [{condition}]
{statement}
{statement}
...
ENDWHILE

Examples:

```
WHILE [P20=0] G4 P1

WHILE [P7500<100]
G90
G0 X0 Y0
Y5
G4 P1
P7500=P7500+1
ENDWHILE
```

See also user variable description for list of accessible / usable variables.

GOTO{data}

{data} is an integer constant or expression with a value from 0 to 99,999

This command causes the motion program execution to jump to the line label (**N**) specified in **{data}**, with no jump back.

If **{data}** is a constant, the path to the label will have been linked before program run time, so the jump is very quick. If **{data}** is a variable expression, it must be evaluated at run time, and the appropriate label then searched for. The search starts downward in the program to the end, and then continues (if necessary) from the top of the program down.

If the specified line label is not found, the program will stop, and the coordinate system's Run-Time-Error bit will be set.

Examples:

```
IF[P7500>0]
GOTO750
```

ENDIF
N740
N750

GOTO[P7500]

See also user variable description for list of accessible / usable variables.

Manual Pallet Changing

The Pallet mounting system is designed to be very repeatable. This allows pallets to be removed, cleaned, inspected, etc. and then placed back on the machine in the same position without a loss of positioning repeatability.

The pallets are held in place an Erowa pallet mounting system as shown in Figure 7.17. The holding force is provided by a pneumatic chuck. This system allows each pallet to mount in the same position every time it is placed on the 363-S.

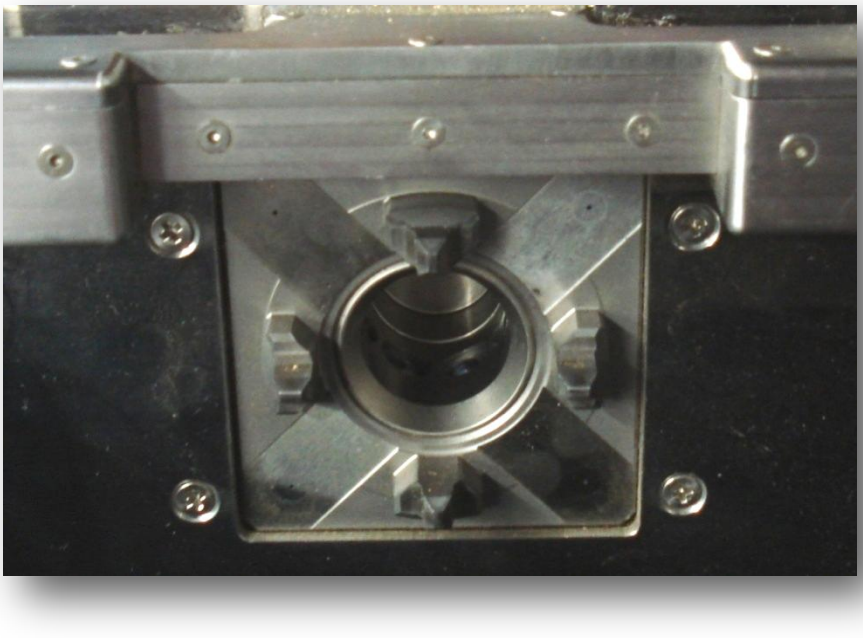


Figure 7.17: Pallet attachment point

Removing the pallet

1. Jog the Z stage away from the pallet at least 40mm.
2. Turn off the spindle.
3. Verify that the tool is clear of the part and pallet. Also verify that the spindle is off.
4. Open the protective lid.
5. Grab the pallet to prevent it from falling when the pneumatic chuck is released. Press the release pallet button in the MDI Tab.
6. Slowly pull the pallet up and out of the working area.

Warning: Tool bits are sharp. Never touch the tool bit.

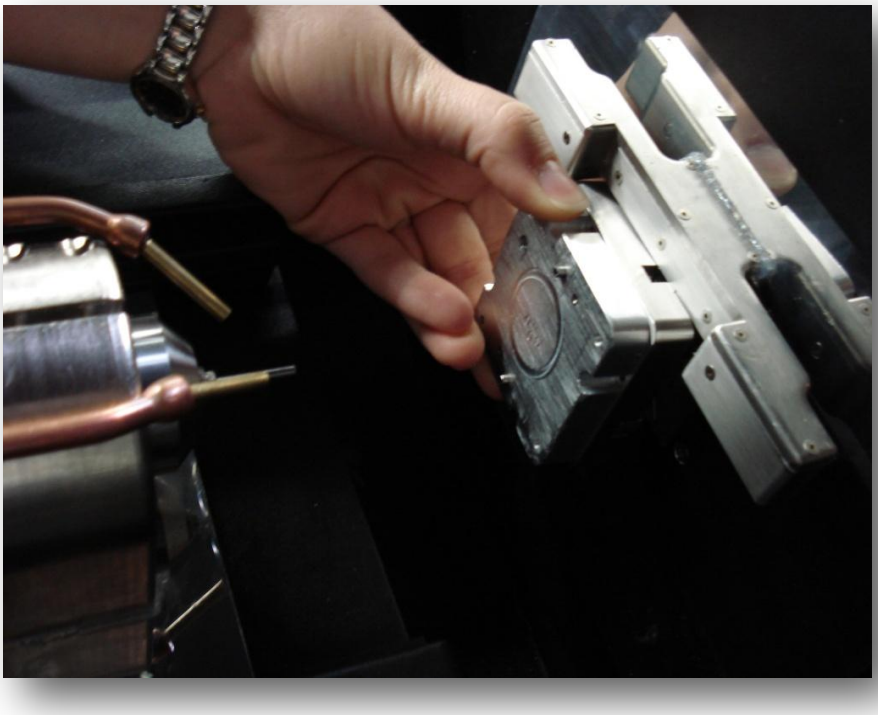


Figure 7.18: Pallet and machinable fixture plate during removal/replacement of pallet

Installing the pallet

1. Move tool and spindle away from pallet mounting area.
2. Disable spindle.
3. Open protective lid.
4. Hold pallet in front of mounting area with spigot placed inside the pneumatic chuck.
5. The round notch (shown in Figure 5.3) on the pallet should face the floor.
6. Press the Clamp Pallet button in the MDI Tab.
7. Check that the pallet is seated firmly and does not move with light pressure.

Warning: Tool bits are sharp. Never touch the tool bit.

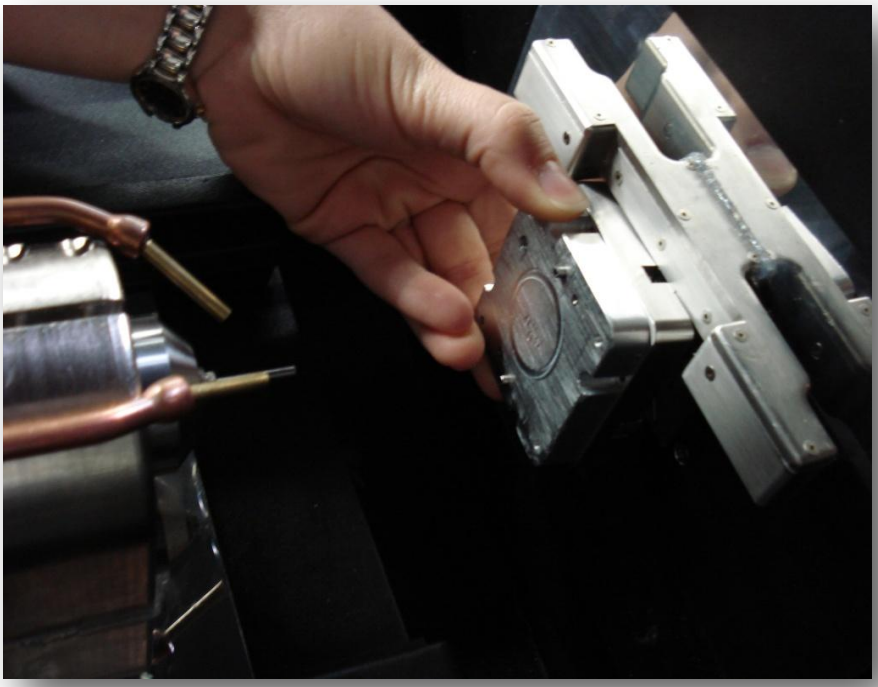


Figure 7.19: Pallet and machinable fixture plate during removal/replacement of pallet

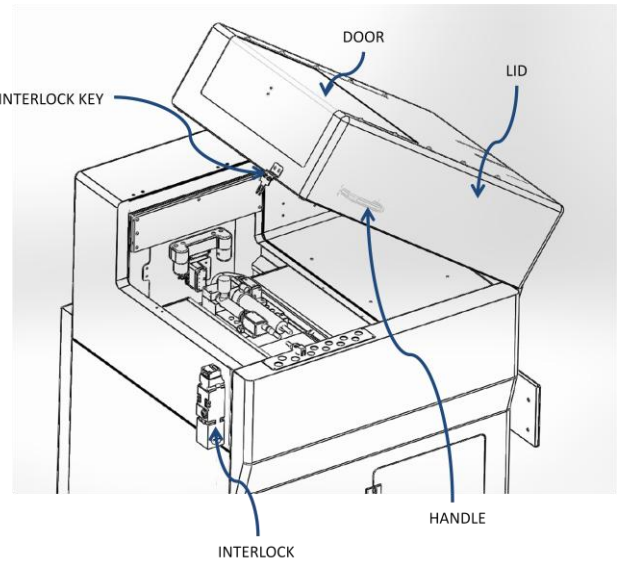
Automatic Part Loading

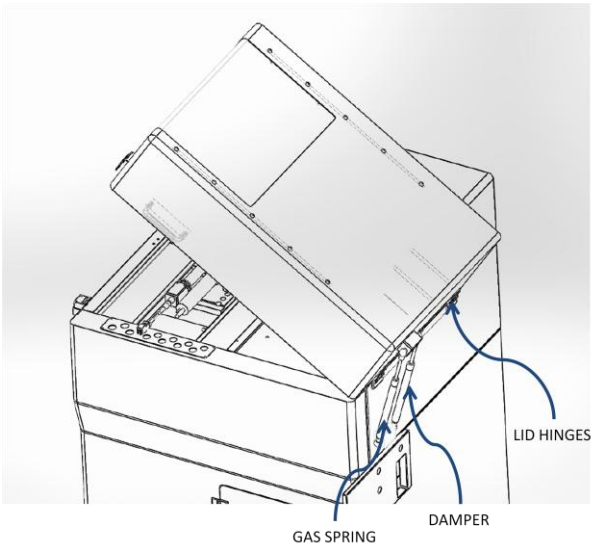
An actuated lid and I/O interface options are available to facilitate automated part loading. This section describes these options and their usage.

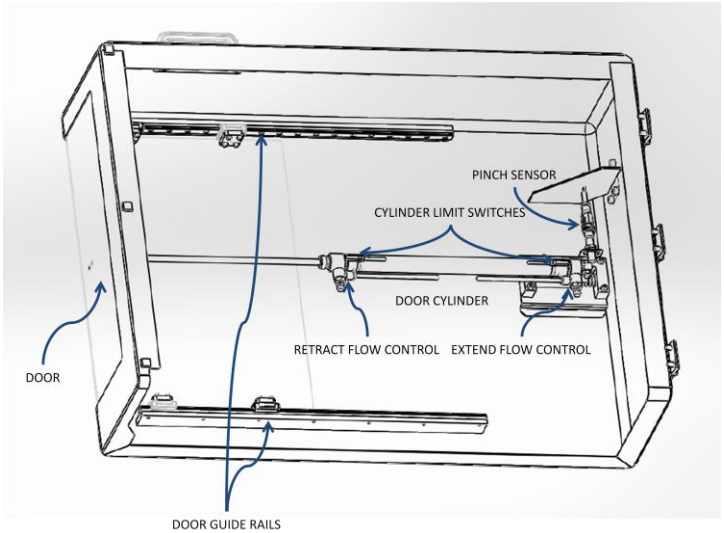
Manual Operation of Robot Lid and Door

The robot lid has both manual and automatic entry. To manually access the machining area, ensure the interlock is unlocked and lift the lid upward and to the right using the handle. If the power to the machine is off, the interlock can be unlocked by turning the toggle on the interlock; however, the machine will not run with the interlock unlocked.

Automatic entry is achieved through the clear robot door. Buttons in the User Tab in the Microlution software interface can be used to actuate the clear door open to the right or closed to the left. Do not attempt to force the automatic door by hand. A pinch sensor detects an error condition when the door closing action is blocked.

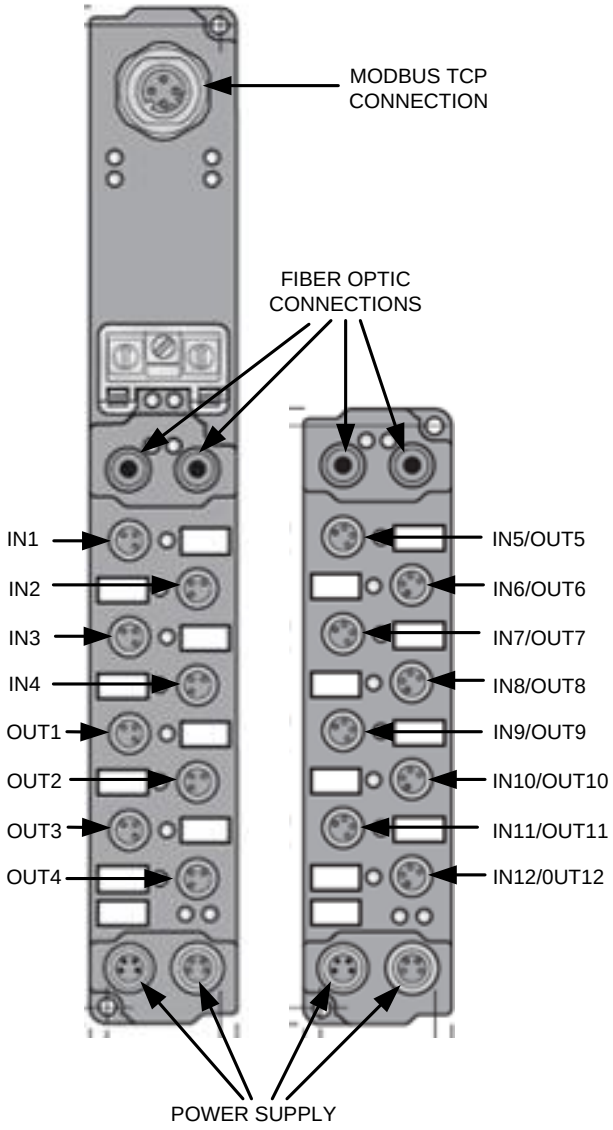




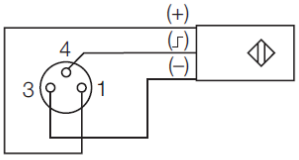


Robot I/O Interface

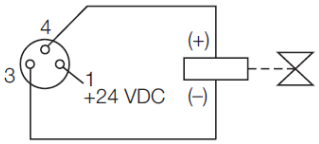
Communication between the machine and the robot is handled using a Turck Modbus TCP I/O module located on the back of the machine. This module contains the capability for 12 digital inputs and 12 digital outputs and is described in the figure below.



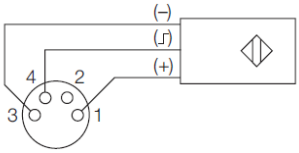
IN1 THRU IN4 WIRING



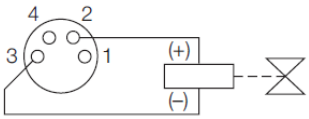
OUT1 THRU OUT4 WIRING



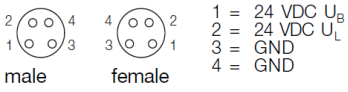
IN5 THRU IN12 WIRING



OUT5 THRU OUT12 WIRING



POWER SUPPLY WIRING



The following table shows the specifications for the I/O module.

TURCK I/O MODULE SPECIFICATIONS

Power supply	
Supply voltage	24 VDC (21.6...28.8 VDC)
Load supply	24 VDC (21.6...28.8 VDC)
Current consumption	
Operating current	75 mA
Digital inputs	8 inputs acc. to EN 61131-2
Input voltage	24 VDC (19...29 VDC)
Supply current	500 mA, short-circuit protected
Switch point "0" / "1"	-3...5 VDC/11...30 VDC
Input filter	3.0 ms
Max. input current	< 7 mA
Digital outputs	8 outputs acc. to EN 61131-2
Output voltage	24 VDC (20...29 VDC)
Output current per channel	500 mA
Load type	ohmic, inductive, lamp load
Short-circuit protection	cyclic
Switching frequency	500 Hz
Load factor	100 %
Galvanic isolation	channels/operating voltage: no; between channels: no; operating voltage/fieldbus: depending on bus system
LED indications	
Bus	function: green – no communication: red
Module status	OK: green – module error: red
Inputs	ON: 8 x green
Short-circuit	sensor inputs shorted: 1 x red
Outputs	ON: 8 x green

I/O Mapping

The following table shows the variable mapping and recommended usage for the I/O lines.

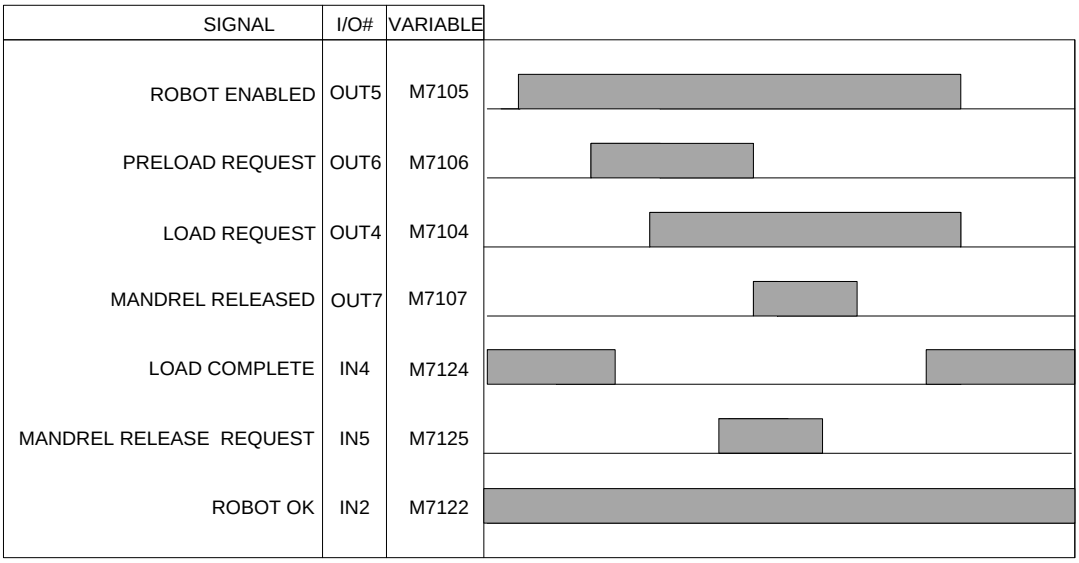
ROBOT I/O MAP

I/O NUMBER	USAGE	M-VARIABLE	DESCRIPTION	M-CODES
OUT1	RESERVED	M7101	RESERVED	RESERVED
OUT2	RESERVED	M7102	RESERVED	RESERVED
OUT3	RESERVED	M7103	RESERVED	RESERVED
OUT4	LOAD	M7104	LOAD SIGNAL	M204.1 (HIGH), M204.2 (LOW)
OUT5	ENABLE	M7105	ENABLE ROBOT (MACHINE OK)	M205.1 (HIGH), M205.2 (LOW)
OUT6	PRELOAD	M7106	PRELOAD REQUEST	M206.1 (HIGH), M206.2 (LOW)
OUT7	PART RELEASED	M7107	PART RELEASED CONFIRMATION	M207.1 (HIGH), M207.2 (LOW)
OUT8	SPARE	M7108	SPARE	M208.1 (HIGH), M208.2 (LOW)
OUT9	SPARE	M7109	SPARE	M209.1 (HIGH), M209.2 (LOW)
OUT10	SPARE	M7110	SPARE	M210.1 (HIGH), M210.2 (LOW)
OUT11	SPARE	M7111	SPARE	M211.1 (HIGH), M211.2 (LOW)
OUT12	SPARE	M7112	SPARE	M212.1 (HIGH), M212.2 (LOW)
IN1	RESERVED	M7121	RESERVED	
IN2	ROBOT ERROR	M7122	HIGH=OK, LOW=ERROR	
IN3	RESERVED	M7123	RESERVED	
IN4	LOAD COMPLETE	M7124	HIGH=COMPLETE, LOW=IN PROGRESS	
IN5	RELEASE PART REQUEST	M7125	HIGH=RELEASE, LOW=CLAMP	
IN6	SPARE	M7126	SPARE	
IN7	SPARE	M7127	SPARE	
IN8	SPARE	M7128	SPARE	
IN9	SPARE	M7129	SPARE	
IN10	SPARE	M7130	SPARE	
IN11	SPARE	M7131	SPARE	
IN12	SPARE	M7132	SPARE	

Automatic Part Loading Sequence G201

A sequence for automatically loading parts is included with the machine. This cycle is called by placing a G201 command in a program. The I/O timing for this sequence and a written description of the sequence is provided below.

G201 TIMING CHART



G201 SEQUENCE DESCRIPTION	
STEP	**DESCRIPTION**
1	SEND LOAD ENABLE SIGNAL TO ROBOT
2	SEND PRE-LOAD SIGNAL TO ROBOT
3	OPEN ROBOT DOOR
4	MOVE MACHINE TO G28 REFERENCE POSITION
5	CHECK MACHINE AT HOME POSITION AND DOOR OPEN
6	SEND LOAD SIGNAL TO ROBOT
7	WAIT FOR MANDREL UNCLAMP REQUEST
8	MANDREL UNCLAMPED, PRE-LOAD SIGNAL RELEASED, MANDREL UNCLAMPED SIGNAL SET TO HIGH
9	WAIT FOR MANDREL CLAMP REQUEST
10	MANDREL IS CLAMPED, MANDREL UNCLAMPED SIGNAL SET TO LOW
11	WAIT FOR LOAD COMPLETE SIGNAL
12	CLOSE ROBOT DOOR
13	RESET OUTPUTS

Tool Setting - Camera

The zero plane for the Z axis can be set manually to any surface of the part or fixture. It is useful to set the surface of the workpiece as the zero plane for the Z axis because most machining operations and programs assume that the surface of the workpiece is the zero plane.

The following is a step by step list of instructions that will guide the user through the process of setting the zero plane for the Z axis.

1. Start with the desired tool in the spindle.
2. Make sure the tool is clear of the part.
3. Move the X and Y axes to place the surface that will be the zero reference for the Z axis directly in front of the tool.
4. Turn the spindle on.
5. Set the spindle speed to the low end of the tool's recommended speed.
6. Use the handwheel to step in towards the part on the Z axis.
7. Monitor the location of the tool using the camera; illustrated in Figure 7.20.

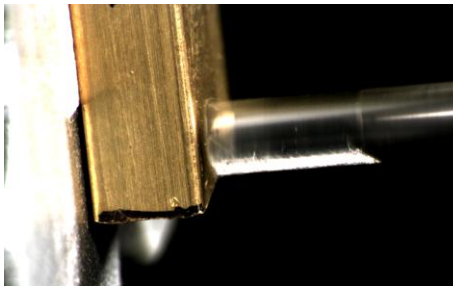


Figure 7.20: Tool near the workpiece surface

8. As the tool gets closer, the X and Y axes can be fine adjusted to make sure the tool is in line with the desired zero reference.
9. Decrease the step size to the smallest setting on the handwheel when the tool is close (<500µm) to the part.
10. Move the tool closer to the material in steps, and check using the camera for the presence of chips around the tool.
11. Watch the camera screen for any chips to indicate that the tool has touched the part, Figure 7.21.

Note: Small tools break easily. Approach increment should be less than 5% of the tool diameter to avoid breaking the tool.

12. As soon as the tool touches the part stop incrementing the Z axis closer.
13. Select only the Z axis on the MDI Tab of the CNC Control Panel.

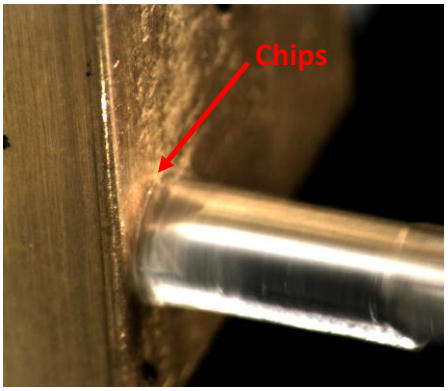


Figure 7.21: Chips created during tool setting

14. Click the 'Set to Zero' button within the MDI Tab to set the current Z location as the program zero.

Note: To set the zero plane using this method the tool will touch the surface and remove some material. If surface finish is critical it may be best to use the fixture as the zero reference and then move the tool out the thickness of the part and reset the zero. After many iterations the fixture will need to be resurfaced because many small holes (<10µm) will have been machined by the multiple zero setting operations.

Note: Be sure to move the tool back far enough to clear the part before zeroing the other axes or starting a program run to prevent tool breakage or part damage.

Hint: By adjusting the lighting so that a shadow is cast by the tool onto the work piece (Figure 7.22), the distance between the tool and its shadow is a good indication of how far the tool is from the work piece. Similarly with the proper camera location in reflective material the tools reflection can be seen in addition to the tool. As the tool gets closer to the surface, the tool and its reflection will get closer. The two will appear to be touching when the tool is touching the surface.

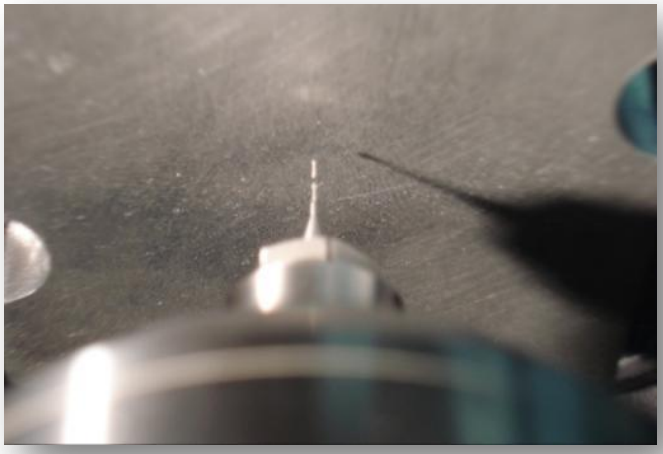


Figure 7.22: The tool and its shadow

Using the Handheld Pendant

1. Select the MDI Tab within the Input Panel of the Main Interface Window.
2. Click the *Enable Handwhee*l Button. This will allow the user to manually adjust each stage along a coordinate axis by using the handheld pendant.
3. Use the dial marked X-Y-Z-4-5-6 to select the axis of movement (Figure 7.23). Currently, the 4, 5, and 6 positions are inactive.
4. Use the increment selection dial to select the handwheel scaling factor (Figure 7.23). X1 moves the selected stage 1 micron per rotation increment, X10 moves the stage 10 microns per rotation increment, and X100 moves the stage 100 microns per rotation increment.
5. To disable manual movement using the handwheel, click the *Disable Handwheel* button.

Figure 7.23: Handheld pendant

Executing a Program

NC programs can be run on the 363-S in three different ways: manually using the Text Editor and the Terminal Panel, Figures 7.24 and 7.25, automatically using the Program Run Tab, and as a part array using the Part Pattern Tab.

Manually Running a Program by Using the Terminal Panel

Warning: The Terminal Panel has the ability to directly address variables and functions in the Delta-Tau CNC controller and therefore should be used with caution. Changes to key operating variables can result in changes in machine accuracy, erratic machine behavior, damage to the machine and/or personal injury. Only personnel specially trained in the operation of the Terminal Panel functionality should use any commands in the Terminal Panel other than the "run" command. The "run" command executes a program. To use the "run" command, type b###r into the Terminal Panel, where ### is the program number. For example, typing b300r would run program 300. Damage to the machine caused by misuse of the Terminal Panel may void the factory warranty. In addition, programs 110-150 are reserved for factory use. Do not use these program numbers.

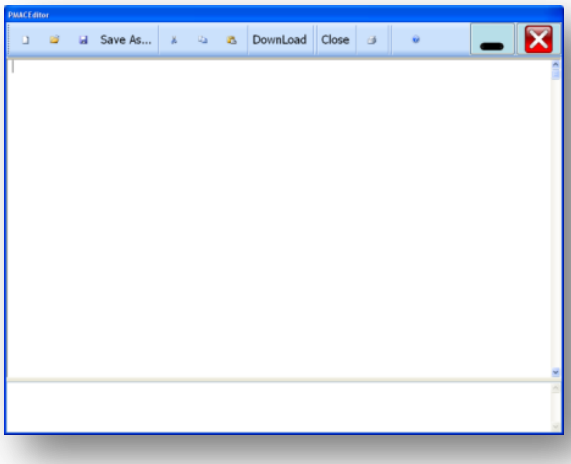


Figure 7.24: Text Editor Panel

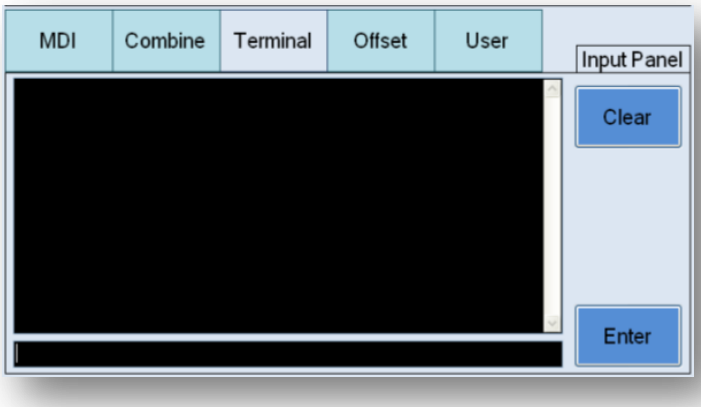


Figure 7.25: Terminal Tab

Programs can be loaded to the controller directly from the built in text editor (Figure 7.24). By saving programs to the controller, they can be run by typing one command in the Terminal panel (Figure 7.25). Programs that contain **loops** or **jump** commands can **only** be executed using the manual method because their entire contents must be in the controller memory in order to perform the jump commands.

Open the text editor using either the *new* or *open* menu buttons. By clicking on the *download* button in the editor (Figure 7.24) the program that is currently in the Editor window will be sent to the controller.

The program will be saved to the file currently open on the 363-S. The first two lines of a program to be downloaded to the 363-S should look like:

close
open prog 300 clear

These lines will close the currently open program, open program 300 and clear anything that is already stored in that location. The remaining lines of the program are then stored in program 300. Any number less than 1000 can be used as the program number.

Note: The file size should be less than 300kb, larger programs can still be run on the 363-S but must utilize the rotary buffer in the CNC Control Panel. For programs larger than 300kb always use the CNC control panel's Program Run Tab.

After downloading the program to the controller, set the desired spindle speed and start the spindle (refer to the Program Run Tab segment of this manual). Run the program by opening the Terminal Tab and type the "run" command into the Terminal Entry Line. The "run" command is executed by typing b###r into the Terminal Entry Line, where ### is the program number. For example, typing b300r would run program 300. Any program loaded into the controllers memory can be run using the Terminal Tab. The program can be stopped at any time using the *Stop* button.

Automatically Running a Program by Using the Combine Tab and the Program Tab

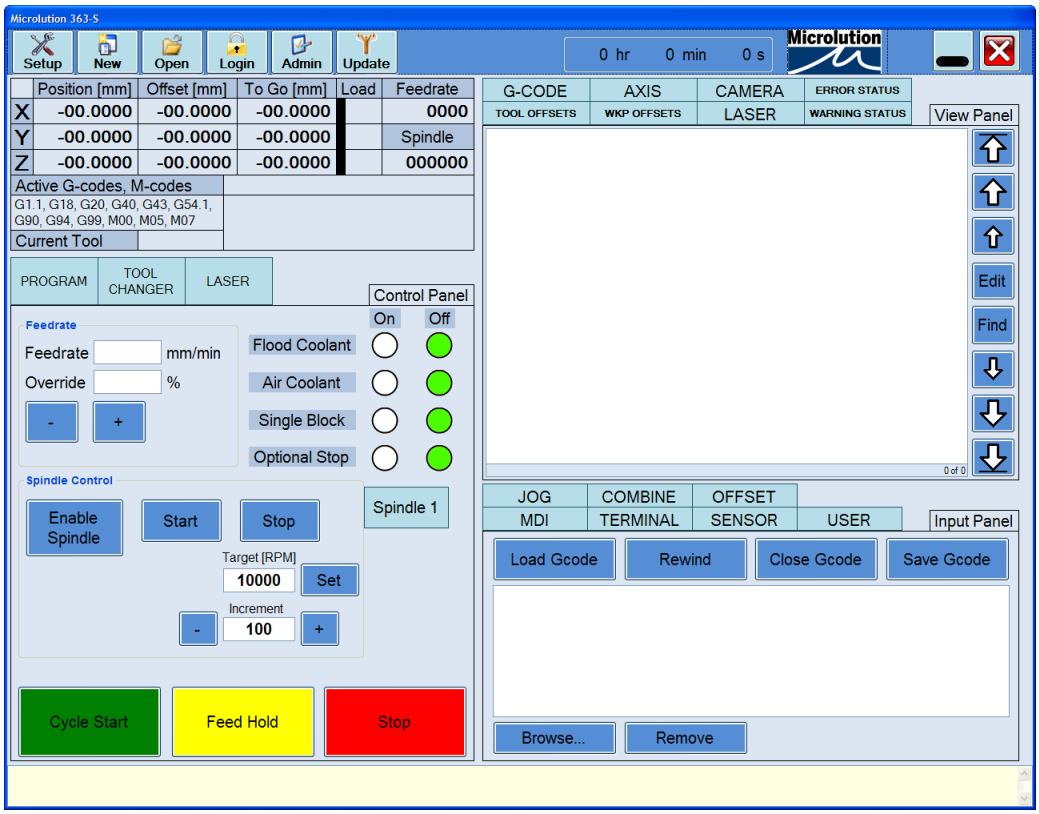


Figure 7.26: Combine Tab and Program Run Tab within the Main Interface Window

The Combine Tab allows larger NC programs to be loaded into memory on the PC and transferred to the controller using a rotary buffer. There is no program size limit when running a program this way.

The Combine Tab is accessible in the Input Panel of the Main Interface Window. The Combine Tab is shown in Fig. 7.26. With the tab open, an NC program can be loaded into the program queue by clicking the *Browse…* button and selecting the appropriate file from the dialog box. After the NC program has been loaded into the queue, clicking the *Load* button will load the program into the rotary buffer and it will consequently appear in the program box in the G-Code Tab of the View Panel (shown in Fig. 7.26).

Note: Large programs may take a few moments to display.

Once the program is loaded, program execution can be controlled through the Program Tab of the Control Panel. If necessary, air and flood coolant can be turned on and off by clicking their corresponding toggle buttons. Single block mode and optional stop mode can be toggled on and off using toggle buttons as well. The spindle can be enabled and its speed adjusted before or during program execution. The feedrate can also be scaled during program execution using the + or - override buttons.

Start the program by clicking the *Cycle Start* button, the program will stop when it reaches the end of the file and/or the *Pause* or *Stop* buttons are pressed.

Part Offsets

Part offsets are recordable program home locations. There are 6 program home locations that can be saved to G-code commands. This feature may be useful if there are multiple common parts that are run on the 363-S and each has a different program home location. Each part's home location could be set as one of the stored G-code home commands. Before running any programs for a particular part, the saved home location could be recalled by selecting it and clicking the *Apply Saved* button. This decreases the setup time for common NC programs.

Save Current Location to a File

1. Open the Jog Tab, Figure 7.30, and click on *Set to Zero*. This will make the current location the program home origin. This will also update the active offset shown in the Dashboard.
2. Open the Offset Tab (Figure 7.31).
3. From the drop down box select the desired save location (G54-G59).
4. Click *Save*. This will take the active program origin and save it to the selected G-code offset.
5. This process can be repeated for a total of up to 6 offsets.
6. To save all six offsets to a file, click the *Save To File* button and type the name of the file in the dialog box that appears.

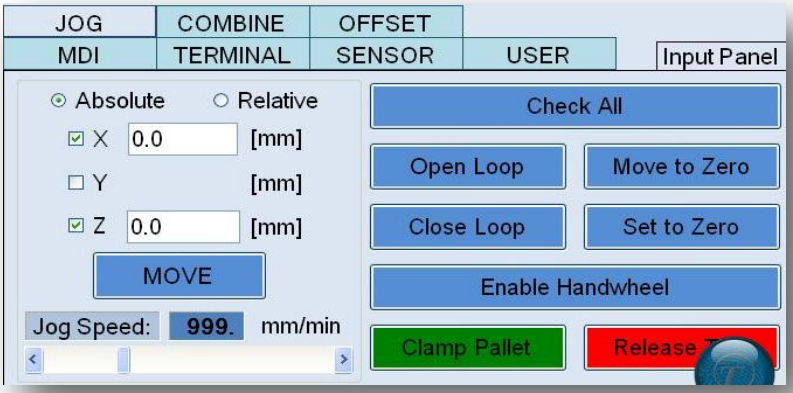


Figure 7.30: Jog Tab

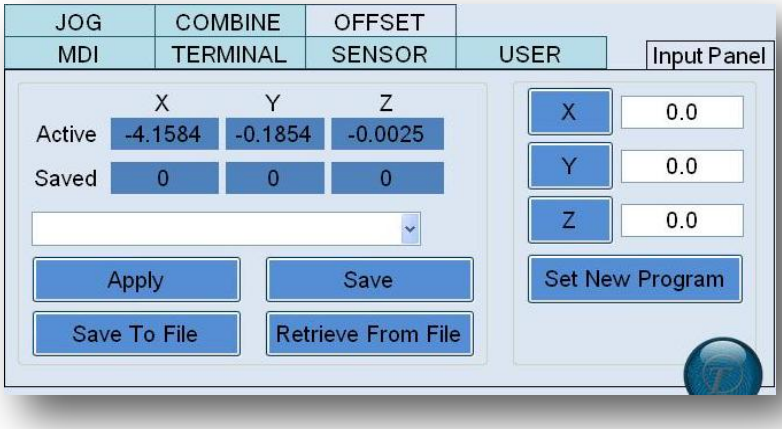


Figure 7.31: Offset Tab

All 6 offsets will be saved to a tab delimited file that can be read back from the file later. Saving to a file is the only way to save the program offsets when the machine is powered down. When the machine is powered up, all offsets are set to their default values of 0.

> **Note:** Shutting down the Microlution 363-S CNC Control Software does not delete the current saved offsets. The offsets that were saved before the program was shut down will be restored the next time the program is started.

Recall Saved Location from a File

1. Click the *Retrieve From* button in the Offset Tab of the software (Figure 7.31).
2. Select the file where the offset locations are stored.
3. The 6 saved offsets will be loaded into the 6 G-codes. These offsets can be previewed by selecting them from the drop down box and the origin will display in the text boxes above the drop down box.
4. Apply the desired offset by selecting the desired drop down box and clicking the *Apply Saved* button.

Set Known Location as Program Origin

1. Within the Offset Tab, Figure 7.31, click the *X*, *Y* and/or *Z* buttons in the middle of the tab to select the axes whose origin should be reset.
2. Type the origin into the text boxes next to the selected axes.
3. Click the *Set New Program* button to make the new origin active. This will update the active origin shown in the Dashboard and in the Offset Tab.
4. If this origin should be saved, select the desired save location from the drop down box (G54-G59).
5. Click *Save Active Origin*. This will take the current program origin and save it to the selected G-code offset.
6. This process can be repeated for a total of up to 6 offsets.
7. Click the *Save To File* button and type the name of the file in the dialog box that appears.

CLEANING/MAINTENANCE

Proper cleaning and maintenance is essential for continued, error free operation of the 363-S. Make sure you read and follow all cleaning/maintenance guidelines before operating the machine.

Additional Machine Guarding to Prevent Dust Ingress

Overview

This section outlines additional dust strip made of felt material that must be added to the machine to prevent plastic dust ingress into the machine. The steps documented below are mandatory for the Optomill (as of 5/20/11) and highly recommended for other 363-S machines that regularly machine plastic. Machines shipped 5/20/11 and later have this dust strip added prior to shipment. Follow the steps below to add or repair this strip.

Required Material

1) 1/8" (3.175mm) thick adhesive-backed felt strip, soft grade (F13 or similar), 1" (25.4mm) wide. This material can be obtained through an industrial supply company (e.g. McMaster-Carr p/n 87555K122) or from Microlution.

Installation procedure

1) Remove the stationary top cover from the machine by removing the (9) M3 fasteners using a Phillips screwdriver.
2) Lay the stationary top cover on a work surface as shown in the figure below. Cut the felt strips into three sections, remove the cover layer for the adhesive, and attach the strips to the top cover as shown below to form a perimeter around the U-shaped notch in the part.
3) Re-attach the stationary top cover to the machine. Ensure that all spacers between the top cover and the machine are replaced properly.
4) Move the X/Y-stage around by hand to make sure that the stage still moves freely without significant additional friction. If additional friction is present, contact Microlution for further instructions.

Figure 8.1: Application of adhesive-backed felt strip.

Recommended Air/Vacuum Setup to Prevent Dust Ingress

Overview:

The air blast and vacuum system work together to remove machined particles from the work area and keep them from ingressing into the internals of the machine. The figures below show the proper setup of the sytem. The vacuum hose attachment is attached and sealed to the chip chute. The vacuum nozzle is installed beneath the work zone. This nozzle is held in place using four magnetic feet and can be removed for cleaning / maintenance. Caution must be used when removing the vacuum nozzle to prevent any contact with a sharp tool tip. The air blast line is attached to the spindle mount and MUST be oriented to point downward to the tool tip and toward the vacuum nozzle. This is because the air blast strongly influences the direction of debris. The air blast line can be bent or adjusted achieve the proper air flow direction.

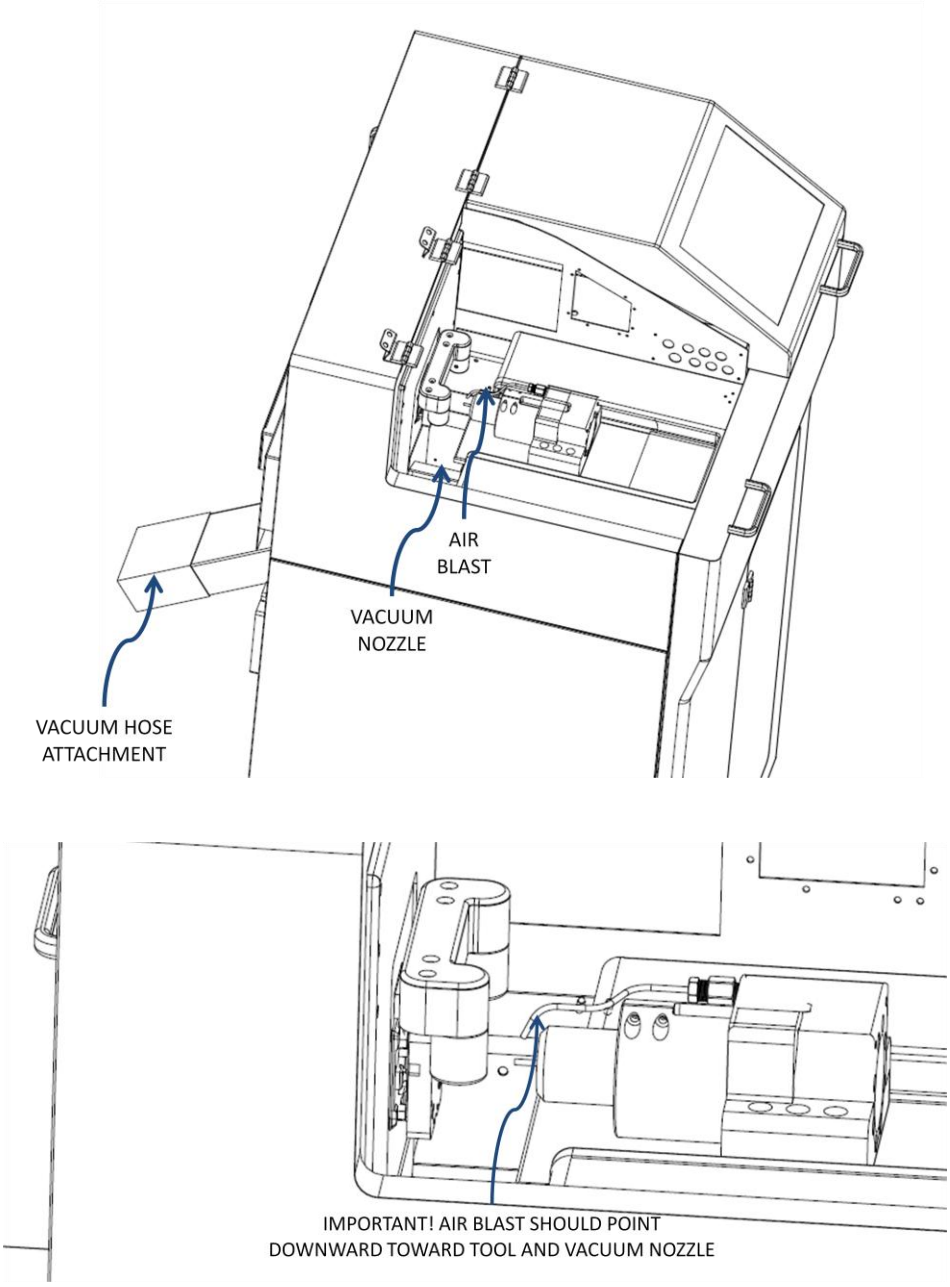


Figure 8.2: Air blast / vacuum setup.

Cleaning

Chip Cleanup

A wet/dry vacuum is the most effective way to remove stray chips in and around the machine. The machine can also be wiped down with a shop towel or fine bristled brush. DO NOT use air blast to blow away chips.

Protective Lid and Chip Hood Cleaning

Clean the clear polycarbonate of the protective lid and chip hood with a mild soap or cleaner.

Maintenance

Preventative Maintenance Schedule			
System	**Description**	**Interval**	**For Additional Information**
Spindle	With air on - wipe spindle with cloth to remove chips	Daily	Spindle User Manual
General	Clean chips from machining zone using brush / vacuum	Daily	Microlution User Manual
Spindle	Remove collet, clean spindle taper, check collet for wear.	Weekly	Spindle User Manual
Pneumatics	Inspect filter bowls for liquid. Drain if necessary	Weekly	Microlution User Manual
Pneumatics	Check that incoming air pressure is 90-100psi	Weekly	Microlution User Manual
Pneumatics	Check tubing for kinks or wear. Replace if worn	Monthly	Microlution User Manual
Cooling Fans	Inspect filters for dust. Clean or replace if dirty	Monthly	Microlution User Manual
General	Dust ingress check. See instructions in User Manual	Monthly	Microlution User Manual

Additional Descriptions for Selected Items

Coolant Filter Inspection

The coolant filter is located inside the coolant system stand. It can be accessed by removing the back cover of the coolant system stand. The coolant filter should be inspected at least once a month under typical use. Under heavy use or in processes that create a large quantity of chips, the filter should be inspected more frequently. The filter can be inspected by simply removing back cover of the stand in looking into the clear filter vessel. If an excessive amount of chips have accumulated within the filter vessel, then the vessel should be cleaned out and the filter replaced.

Filter Replacement

1. Remove the back cover of the coolant system stand.
2. Loosen the screws that hold the coolant filter vessel to the back of the stand.
3. Remove the coolant filter vessel from the stand with both coolant lines still attached.
4. Unplug the coolant line from the output of the filter vessel. Place a receptacle underneath the vessel while doing this to catch any coolant that flows out of the line or vessel.
5. Pour some of the coolant out of the filter vessel and into the receptacle through the output port. Pour until you can see the level of the coolant through the clear vessel.
6. Unscrew the clear part of the filter vessel from the top.
7. Remove the old filter.
8. Dump out the rest of the coolant that is inside the vessel and clean out any old chips.
9. Insert a new filter into the vessel (contact Microlution for order information about replacement filters).
10. Reattach the bottom of the filter vessel to the top and plug in the output coolant line.
11. Turn on the flood coolant through the software to test for any leaks in the housing. It may take a moment for the coolant to start flowing out of the coolant nozzle while the filter vessel refills.
12. If there are no leaks, reattach the filter housing to the stand and replace the back cover.

Coolant Change

The coolant should be replaced at least as often as the filter bag. If the coolant begins to get foamy and/or begins to smell bad, the coolant should be changed independently. To change the coolant, follow these instructions:

1. Remove the filter vessel from the stand (follow steps 1-3 in the Filter Replacement section above) so that both drain ports can be accessed.
2. Place a pan or bucket inside the stand under one of the two drain ports.
3. Turn the valve on the drain port that the bucket is under to allow coolant to flow out of the tank.
4. When all the coolant has drained, close the valve and repeat steps 2 and 3 for the other port.
5. New coolant can be poured into the top of the tank.
6. Replace filter vessel and back cover of the coolant system stand.

Coolant Pump Maintenance

If the coolant pump begins to leak or you notice a drop in performance, contact Microlution for assistance and/or replacement.

Electrical Panel Fuses

If a fuse burns out on the electrical panel, it will need to be replaced. This can be done by removing the left side panel and accessing the power distribution box. The power distribution box is on the middle-left of the electronics panel and has a clear cover. There are 4 fuses. F1 is 10A fast acting. F2A and F2B are 5A slow-blow type fuses. F3 is 3A slow-blow fuse.

CAUTION: Ensure machine is off and that there is no power flowing to the power distribution box before replacing fuses.

Regular Dust Ingress Check

Overview

This section describes an update to the machine preventative maintenance for the Optomill to check for dust ingress into the X/Y stage. This check should be performed monthly to ensure error free operation.

Procedure

1) Power down machine.
2) Thoroughly clean machining area to remove plastic dust using a brush and vacuum.
3) Remove the fixed top cover on the X/Y stage (see figure below) using a Phillips screwdriver.
4) Remove the moving mirror plate on the X/Y stage (see figure below) using a Phillips screwdriver.
5) Inspect internal components of X/Y stage for dust ingress. Record observations and take pictures for future reference. If significant dust is observed, contact Microlution.
6) Replace mirror plate and fixed top cover.
7) Remove machine side panels (see figure below) and inspect machine cabinet for dust. Record observations and take pictures for future reference. Vacuum any dust observed in machine cabinet.
8) Replace machine side panels.

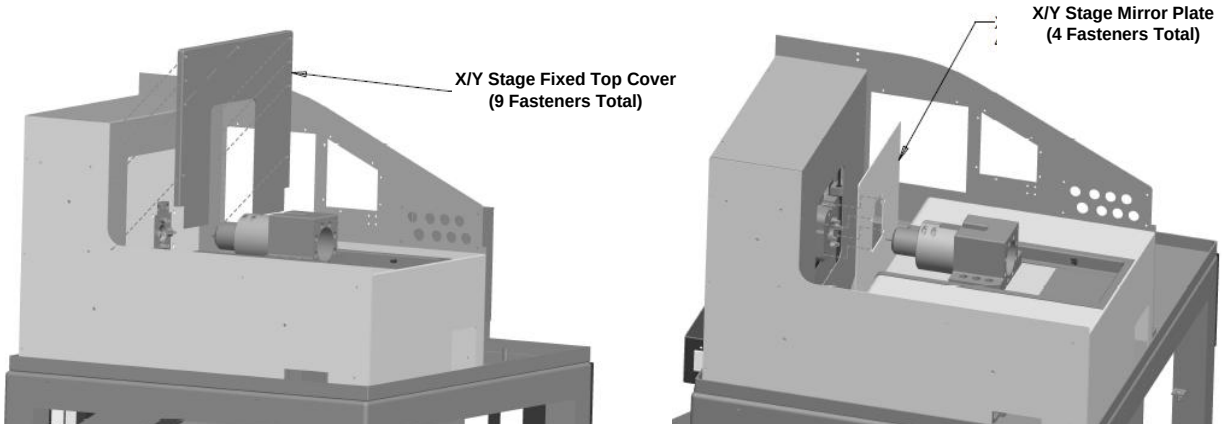


Figure 8.3: X/Y stage chip guard removal.

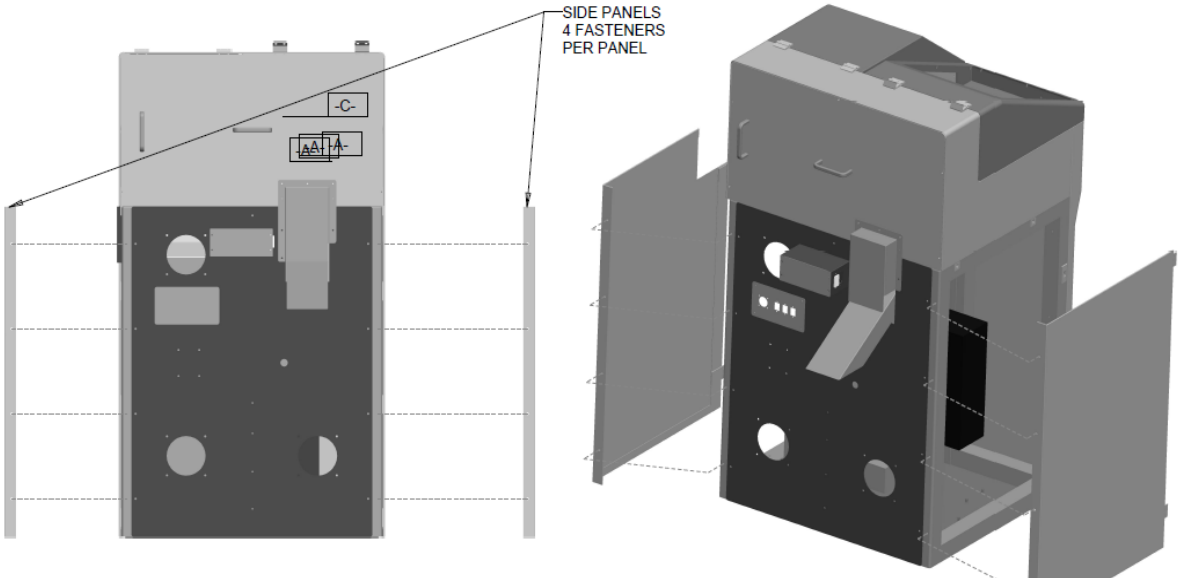


Figure 8.4: Machine side panel removal.

SUPPORTED G AND M CODES

Table 9.1: Supported G and M Codes

Code	Description	Parameters
G00	Rapid positioning	X Y Z
G01	Linear interpolation	X Y Z F
G02	CW circular interpolation	X Y Z I J K F or X Y Z R F
G03	CCW circular interpolation	X Y Z I J K F or X Y Z R F
G04	Dwell	P
G10	Data Setting	L, P, R, X, Y, Z
G11	Data Setting End	
G17	X-Y plane selection	
G18	X-Z plane selection	
G19	Y-Z plane selection	
G20	Set units to inches	
G21	Set units to millimeters	
G28	Return to machine reference position (metric only)	X Y Z
G28.1	Machine at reference position check	
G40	Tool radius compensation off	
G41	Tool radius compensation left	D
G42	Tool radius compensation right	D
G43	Apply Tool Offset	F H Z
G49	Remove Tool Offset	F H Z
G54-G59	Work Offsets One Through Six, Additional	
G65	Macro Call	A-M, P-Z
G80	Canned cycle cancel	
G81	Drilling spot – drill	X Y Z R F
G82	Drilling cycle, counter bore	X Y Z R P F
G83	Drilling cycle, peck	X Y Z R Q F
G90	Absolute Positioning	
G91	Incremental Positioning	
G98	Canned cycle Initial Level return (default)	
G99	Canned cycle “R” return level	
G120	Tool length/diameter measurement *(option required)*	B H F I J
G121	Tool breakage detection *(option required)*	H L S
G123	Machine Workpiece Measurement *(option required)*	H J W X Y Z
G201	Pallet Change *(option required)*	
M00	Programmed Stop	
M01	Optional Stop	
M02	End of Program	
M03	Spindle Clockwise	S
M05	Spindle Off	
M06	Tool Change *(option required)*	T
M07	Air Coolant On	
M08	Cutting Fluid On *(option required)*	
M09	Cutting Fluid / Air Coolant Off	

M30	End of Program with Rewind	
M98	Subroutine Call	L, P
M99	End Macro/Subroutine Call	
M101	Pallet Clamp	
M102	Pallet Unclamp	
M103	Machine Light On *(option required)*	
M104	Machine Light Off *(option required)*	
M106	Tool Guard Retract *(option required)*	
M107	Tool Guard Extend *(option required)*	
M110.1	Robot Door Open *(option required)*	
M110.2	Robot Door Close *(option required)*	
M110.3	Robot Door is Open Check *(option required)*	
M110.4	Robot Door is Closed Check *(option required)*	
M112.1	Mandrel Clamp *(option required)*	
M112.2	Mandrel Unclamp *(option required)*	
M112.3	Mandrel is Clamped Check *(option required)*	
M112.4	Mandrel is Unclamped Check *(option required)*	
M201	Robot Pulse Sync *(3R robot option required)*	
M202	Robot Pulse Clock *(3R robot option required)*	
M203.1	Robot Data On *(robot option required)*	
M203.2	Robot Data Off *(robot option required)*	
M204.1	Robot Load On *(robot option required)*	
M204.2	Robot Load Off *(robot option required)*	
M205.1	Robot Enable Load On *(robot option required)*	
M205.2	Robot Enable Load Off *(robot option required)*	
M206.1	Robot Pre Load On *(robot option required)*	
M206.2	Robot Pre Load Off *(robot option required)*	
M207.1	Robot Part Released Output On *(robot option required)*	
M207.2	Robot Part Released Output Off *(robot option required)*	
M208.1	Robot Output 8 On *(robot option required)*	
M208.2	Robot Output 8 Off *(robot option required)*	
M209.1	Robot Output 9 On *(robot option required)*	
M209.2	Robot Output 9 Off *(robot option required)*	
M210.1	Robot Output 10 On *(robot option required)*	
M210.2	Robot Output 10 Off *(robot option required)*	
M211.1	Robot Output 11 On *(robot option required)*	
M211.2	Robot Output 11 Off *(robot option required)*	
M212.1	Robot Output 12 On *(robot option required)*	
M212.2	Robot Output 12 Off *(robot option required)*	

Common G – Codes, M – Codes and their syntax

G00 Rapid Move Mode

Format: G00 [X.. Y.. Z..]

Required Inputs:

Optional Inputs:

		Default:
X	Move destination along X-axis	current location
Y	Move destination along Y-axis	current location
Z	Move destination along Z-axis	current location

G01 Linear Move Mode

Format: G01 [X.. Y.. Z.. F..]

Required Inputs:

Optional Inputs:

		Default:
X	Move destination along X-axis	current location
Y	Move destination along Y-axis	current location
Z	Move destination along Z-axis	current location
F	Feedrate	previously used feedrate

Note: G01 can be used in conjunction with G43 in specialized format below:
G01 G43 Z.. H.. F..

G02 Circular Move Mode Clockwise

Format: G02 X.. Y.. [F.. I.. J..] or G02 X.. Y.. [F.. R..]

Required Inputs:

X	Move destination along X-axis
Y	Move destination along Y-axis

Optional Inputs:

F	Feedrate
I	Arc move along the X-axis from current location (circle center) to start
J	Arc move along the Y-axis from current location (circle center) to start
R	Circle radius

Note: This command will not work if there is no I, J or R specified. Please use one of the formats listed above. A single I or J input will also work. There are no defaults for this command.

G03 Circular Move Mode Counter Clockwise

Format: G03 X.. Y.. [F.. I.. J..] or G03 X.. Y.. [F.. R..]

Required Inputs:

X	Move destination along X-axis
Y	Move destination along Y-axis

Optional Inputs:

F	Feedrate
I	Arc move along the X-axis from current location (circle center) to start
J	Arc move along the Y-axis from current location (circle center) to start
R	Circle radius

Note: This command will not work if there is no I, J or R specified. Please use one of the formats listed above. A single I or J input will also work. There are no defaults for this command.

G04 Dwell

Format: G04 P.. or G04 X..

Required Inputs:

P	Dwell time in seconds

Optional Inputs:

		Default:
X	Dwell time in seconds	0 seconds

G10 Data Setting

Format: G10 L.. or G10 L.. P.. [X.. Y.. Z..] or G10 L.. P.. R..

Required Inputs:

	L	Data setting group number

Optional Inputs:

	P	Data register
	R	Data value
	X	X-axis offset
	Y	Y-axis offset
	Z	Z-axis offset

Note: This command will only work in one of the above formats. There are no defaults for this command.

Note: If data setting is done while in absolute mode, the current offset will be over-written. If data setting is done while in incremental mode, the value will be added to the current offset.

G11 Data Setting Modal End

Format: G11 []

Required Inputs:

Optional Inputs:

G17 X-Y Plane Designation

Format: G17 []

Required Inputs:

Optional Inputs:

G18 Z-X Plane Designation

Format: G18 []

Required Inputs:

Optional Inputs:

G19 Y-Z Plane Designation

Format: G19 []

Required Inputs:

Optional Inputs:

G20 Metric to English (mm to in) Unit Conversion

Format: G20 []

Required Inputs:

Optional Inputs:

G21 English to Metric (in to mm) Unit Conversion

Format: G21 []

Required Inputs:

Optional Inputs:

G28 Return to Machine Reference Position

Format: G28 [X.. Y.. Z..]

Required Inputs:

Optional Inputs:

			Default:
	X	Move destination along X-axis	current location
	Y	Move destination along Y-axis	current location
	Z	Move destination along Z-axis	current location

Note: The recommended form is listed below.

G91 G28 Z0
G91 G28 X0 Y0

G28.1 Machine at Reference Position Check
Format: G28.1 []
Required Inputs:
Optional Inputs: Default:

G40 Tool Radius Cutter Compensation Off
Format: G40 []
Required Inputs:
Optional Inputs:

G41 Tool Radius Cutter Compensation Left
Format: G41 D..
Required Inputs:
D Tool number
Optional Inputs:

G42 Tool Radius Cutter Compensation Right
Format: G42 D..
Required Inputs:
D Tool number
Optional Inputs:

G43 Tool Length Offset Apply
Format: G43 H.. [F.. Z..]
Required Inputs:
H Tool number
Optional Inputs: Default:
F Feedrate previously used feedrate
Z Move destination along Z-axis current location
Note: This offset is in addition to the current offset. It will take into account the previously applied offset, not overwrite them.
Note: G43 can be used in conjunction with G01 in specialized format below:
G01 G43 Z.. H.. F..

G49 Tool Length Offset Remove
Format: G49
Required Inputs:
Optional Inputs:
Note: This command will only work if the last offset commanded was a tool length offset, G43.

G54 Workpiece Offset 1
Format: G54 []
Required Inputs:
Optional Inputs:
Note: This offset will overwrite the current offset.

G54.1 Additional Workpiece Offsets
Format: G54.1 P..
Required Inputs:
P Additional Workpiece Offset Number

Optional Inputs:
Note: This offset will overwrite the current offset. P parameter inputs can only be 1 – 394. Any others will be discarded and no offset will be applied.

G55 Workpiece Offset 2
Format: G55 []
Required Inputs:
Optional Inputs:
Note: This offset will overwrite the current offset.

G56 Workpiece Offset 3
Format: G56 []
Required Inputs:
Optional Inputs:
Note: This offset will overwrite the current offset.

G57 Workpiece Offset 4
Format: G57 []
Required Inputs:
Optional Inputs:
Note: This offset will overwrite the current offset.

G58 Workpiece Offset 5
Format: G58 []
Required Inputs:
Optional Inputs:
Note: This offset will overwrite the current offset.

G59 Workpiece Offset 6
Format: G59 []
Required Inputs:
Optional Inputs:
Note: This offset will overwrite the current offset.

G65 Macro Call
Format: G65 P.. [A.. B.. C.. D.. E.. F.. G.. H.. I.. J.. K.. L.. M.. Q.. R.. S.. T.. U.. V.. W.. X.. Y.. Z..]
Required Inputs:
P Macro Program Number
Optional Inputs:
A-M, Q-Z Macro Parameters

G80 Canned Cycle Cancel
Format: G80 []
Required Inputs:
Optional Inputs:

G81 Drilling Canned Cycle
Format: G81 X.. Y.. R.. Z.. F.. K..
Inputs:
X Center location of hole along X-axis
Y Center location of hole along Y-axis
R Clearance plane of Z-axis
Z Total depth of tool point

F Feedrate
K Number of repeats

G82 Spot-Drilling Canned Cycle
Format: G82 X.. Y.. R.. Z.. P.. F.. K..
Inputs:
X Center location of hole along X-axis
Y Center location of hole along Y-axis
R Clearance plane of Z-axis
Z Total depth of tool point
P Dwell in seconds at the bottom of the drill cycle
F Feedrate
K Number of repeats

G83 Deep hole Drilling Canned Cycle
Format: G83 X.. Y.. R.. Z.. P.. Q.. I.. J.. K.. F.. K..
Inputs:
X Center location of hole along X-axis
Y Center location of hole along Y-axis
R Clearance plane of Z-axis
Z Total depth of tool point
P Dwell in seconds at the bottom of the drill cycle
F Feedrate (mm/min)
Q Incremental distance of each peck
I Depth of first peck
J Peck reduction for each peck after first
K Number of repeats

G90 Absolute Mode
Format: G90 []
Required Inputs:
Optional Inputs:

G91 Incremental Mode
Format: G91 []
Required Inputs:
Optional Inputs:

G98 Canned Cycle Initial Level Return
Format: G98 []
Required Inputs:
Optional Inputs:

G99 Canned Cycle "R" Level Return
Format: G99 []
Required Inputs:
Optional Inputs:

G120 Tool Sense
Format: G120 B.. H.. [J.. S.. X.. Z..]
Required Inputs:
B Tool sense type
H Tool number

Optional Inputs:			Default:
	J	Additional length offset	0 .0mm
	S	Desired spindle speed	5000 rpm
	X	Radial location of length measurement	0.0mm
	Z	Axial location of diameter measurement	0.5mm

Note: The tool sense type (B) can either be 1 (tool length sense only), 2 (tool length sense with one-sided diameter sense), or 3 (tool length sense with two-sided diameter sense).

G121 Tool Sense Breakage Detection

Format: G121 H.. [L.. S..]

Required Inputs:

	H	Tool number	
Optional Inputs:			Default:
	L	Tolerance for breakage detection	0.1 mm
	S	Spindle speed	5000 rpm

G123 Automatic Workpiece Detection

Format: G123 [H.. J.. W.. X.. Y.. Z..]

Required Inputs:

Optional Inputs:			Default:
	H	Tool number	tool 1
	J	Offset off of the detected location	0 mm
	W	Workpiece offset number	0
	X	Detection location along the X-axis	current location
	Y	Detection location along the Y-axis	current location
	Z	Initial Z-axis move destination	current location

Note: The workpiece offset number can be 0-6. The numbers 1-6 represent the workpiece offsets G54 – G59. The number 0 enables the user to immediately apply without saving it to a particular workpiece offset.

G201 Pallet Change

Format: G201

Required Inputs:

Optional Inputs:

M00 Program Stop Flag

Format: M00

Required Inputs:

Optional Inputs:

M01 Optional Stop Flag

Format: M01

Required Inputs:

Optional Inputs:

Note: The optional stop flag will only stop the program if the optional stop indicator located in the Program Panel is ON.

M02 End of Program Flag
Format: M02
Required Inputs:
Optional Inputs:

M03 Start Spindle Clockwise
Format: M03
Required Inputs:
Optional Inputs:

M05 Stop Spindle
Format: M05
Required Inputs:
Optional Inputs:

M06 Tool Change
Format: M06
Required Inputs:
Optional Inputs:
Note: Before this command, a T command must be issued with the appropriate tool number.
T01 M06

M07 Air Blast Coolant On
Format: M07
Required Inputs:
Optional Inputs:

M08 Flood Coolant On
Format: M08
Required Inputs:
Optional Inputs:

M09 All Coolant Off
Format: M09
Required Inputs:
Optional Inputs:

M30 End of Program Flag, Rewind NC code
Format: M30
Required Inputs:
Optional Inputs:

M98 Subroutine Call
Format: M98 P.. [L..]
Required Inputs:
P Subroutine Number
Optional Inputs:
L Number of Subroutine Repeats

M101 Pallet Clamp
Format: M101
Required Inputs:
Optional Inputs:

M102 Pallet Unclamp
Format: M102
Required Inputs:
Optional Inputs:

M103 Machine Light On
Format: M103
Required Inputs:
Optional Inputs:

M104 Machine Light Off
Format: M104
Required Inputs:
Optional Inputs:

M106 Tool Guard Retract
Format: M106
Required Inputs:
Optional Inputs:

M107 Tool Guard Extend
Format: M107
Required Inputs:
Optional Inputs:

M110.1 Robot Door Open
Format: M110.1
Required Inputs:
Optional Inputs:

M110.2 Robot Door Close
Format: M110.2
Required Inputs:
Optional Inputs:

M110.3 Robot Door is Open Check
Format: M110.3
Required Inputs:
Optional Inputs:

M110.4 Robot Door is Closed Check
Format: M110.4
Required Inputs:
Optional Inputs:

M112.1 Mandrel Clamp
Format: M112.1
Required Inputs:
Optional Inputs:

M112.2 Mandrel Unclamp
Format: M112.2

Required Inputs:
Optional Inputs:

M112.3 Mandrel is Clamped Check
Format: M112.3
Required Inputs:
Optional Inputs:

M112.4 Mandrel is Unclamped Check
Format: M112.4
Required Inputs:
Optional Inputs:

M201 Robot Pulse Sync
Format: M201
Required Inputs:
Optional Inputs:

M202 Robot Pulse Clock
Format: M202
Required Inputs:
Optional Inputs:

M203.1 Robot Data On
Format: M203.1
Required Inputs:
Optional Inputs:

M203.2 Robot Data Off
Format: M203.2
Required Inputs:
Optional Inputs:

M204.1 Robot Load On
Format: M204.1
Required Inputs:
Optional Inputs:

M204.2 Robot Load Off
Format: M204.2
Required Inputs:
Optional Inputs:

M205.1 Robot Enable Load On
Format: M205.1
Required Inputs:
Optional Inputs:

M205.2 Robot Enable Load Off
Format: M205.2
Required Inputs:
Optional Inputs:

M206.1 Robot Pre Load On
Format: M206.1
Required Inputs:
Optional Inputs:

M206.2 Robot Pre Load Off
Format: M206.2
Required Inputs:
Optional Inputs:

M207.1 Robot Part Released Output On
Format: M207.1
Required Inputs:
Optional Inputs:

M207.2 Robot Part Released Output Off
Format: M207.2
Required Inputs:
Optional Inputs:

M208.1 Robot Output 8 On
Format: M208.1
Required Inputs:
Optional Inputs:

M208.2 Robot Output 8 Off
Format: M208.2
Required Inputs:
Optional Inputs:

M209.1 Robot Output 9 On
Format: M209.1
Required Inputs:
Optional Inputs:

M209.2 Robot Output 9 Off
Format: M209.2
Required Inputs:
Optional Inputs:

M210.1 Robot Output 10 On
Format: M210.1
Required Inputs:
Optional Inputs:

M210.2 Robot Output 10 Off
Format: M210.2
Required Inputs:
Optional Inputs:

M211.1 Robot Output 11 On
Format: M211.1
Required Inputs:

Optional Inputs:

M211.2 Robot Output 11 Off
Format: M211.2
Required Inputs:
Optional Inputs:

M212.1 Robot Output 12 On
Format: M212.1
Required Inputs:
Optional Inputs:

M212.2 Robot Output 12 Off
Format: M212.2
Required Inputs:
Optional Inputs:

APPENDIX A – LEGACY / CUSTOM ORDER

This section contains instructions on legacy and specialized components that may be specific to a customer's machine. Please disregard this section if this information does not pertain to your machine.

Spindle Changing (Kinematic Mount Type)

The spindle mounting system is designed to be very repeatable. Additionally, the mounting system allows for quick removal and attachment so that spindles can be cleaned, inspected, changed, etc. The spindle is held in place by two ball and Vee block connections and two ball and split-Vee connections. The balls are located at the four corners of the spindle mount. The two Vees are located closest to the pallet on the Z-axis of the machine as shown in Figure 7.2. The two split-Vees are also located on the Z-axis but are farther from the pallet (See Figure 7.2). The holding force is provided by four permanent magnets. The arrangement of the Vee and split-Vee blocks and the implementation of the rare-earth magnets provide an exact kinematic support mounting system that has sub-micron repeatability.

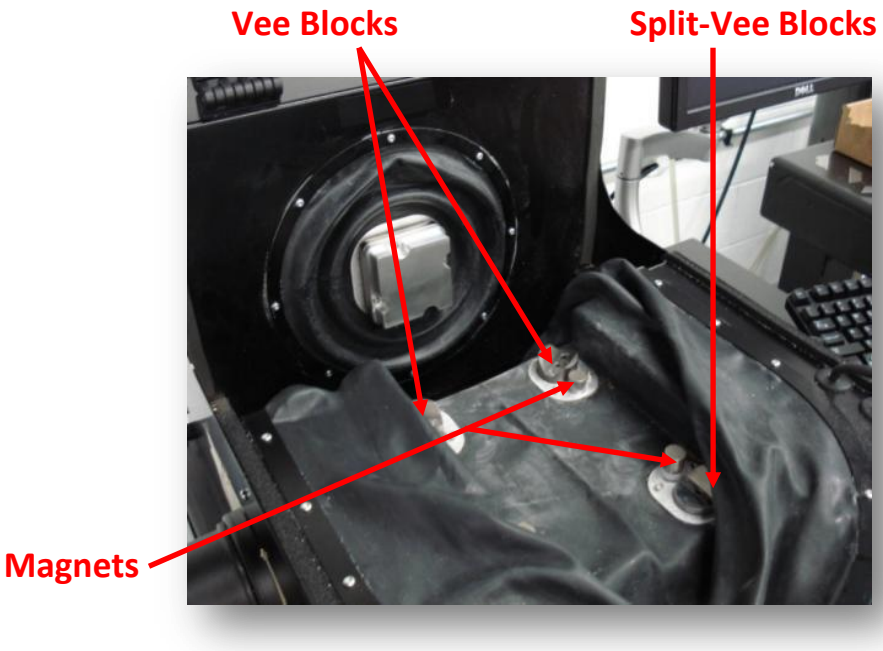


Figure 7.2: Spindle attachment points

Removing the Spindle

1. Jog the Z stage away from the pallet at least 40mm.
2. Turn off the spindle.
3. Verify that the tool is clear of the workpiece and the pallet. Also, verify that the spindle is off.
4. Open the protective lid.
5. Disconnect the air and electrical connections on the spindle if possible.
6. Lift the back of the spindle off the split-Vee blocks as illustrated in Figure 7.3, withdraw the spindle from the working volume, and pull away from the front Vee blocks.

Warning: Magnetic force decreases rapidly as distance increases. Spindle can accidentally be pulled into the work piece, pallet, rubber boot or other parts of the operator or machine when removing the spindle, be careful.

Warning: Tool bits are sharp. Handle tool bits with care.

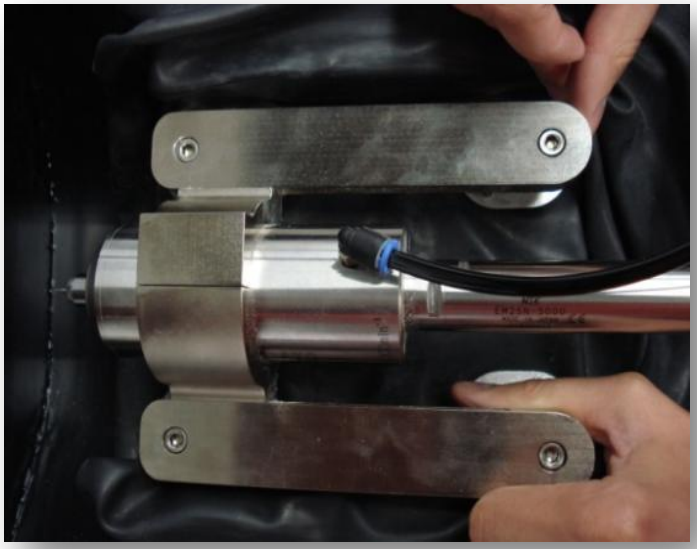


Figure 7.3: Removing the spindle

Installing the Spindle

1. Within the MDI Tab of the CNC control panel, move the Z stage away from the pallet mounting area.
2. Verify that the spindle is disabled within the Program Run Tab of the CNC control panel. Refer to Spindle Operation section for information on the location of the *Disable Spindle* button.
3. Open the protective lid.
4. Hold the spindle above the mounting area with the balls roughly lined up with the Vee blocks.
5. Lower the front balls onto their Vee blocks while holding the spindle at approximately a 30 degree angle (working end pointed down).
6. Once the front balls locate in their Vee blocks as shown in Figure 7.3, lower the back of the spindle into the other blocks.
7. Check that the spindle is seated firmly on the Vee blocks.
8. Attach the spindle's air and electrical connections if necessary. Refer to the supplied spindle user's manual for detailed information on the spindle's air and electrical connections.

Warning: Magnets will pull the spindle toward the attachment point. Keep fingers clear.

Warning: Tool bits are sharp. Never touch the tool bit.

Workpiece sensing

The workpiece sensing function uses a laser displacement sensor attachment to measure the thickness of the workpiece at a particular location.

Manual Operation Mode

Below are the steps to use the workpiece sensor manually using the touch screen interface.

1. Stop spindle if rotating.
2. Remove tool from spindle.
3. Disable spindle using the Disable Spindle button.
4. Remove spindle.
5. Install laser.
6. Move machine to desired X/Y position.
7. Close lid.
8. Press *Workpiece Sensing* button located in the User Tab of the Input Panel.
9. Ensure laser is on. The laser will move toward the workpiece until it reaches the appropriate measurement range. The laser sensing is complete when the laser turns off and the stage retracts.
10. The workpiece thickness offset is automatically saved to the G54 workpiece offset register. The offset will also be displayed in the text box next to the Workpiece Sensing button.

Operation in an NC Program

Workpiece sensing is executed using G122. The syntax for this code is as follows:

G122 Wx

Where x is the worpiece offset number between 1-6 to which the measurement will be saved. If this parameter is set to 1 the measurement is saved to the G54 registers, if set to 2 the measurement is saved to G55 registers, and so on.

Standard implementation within an NC program might look like this (below is a description of each step):

```
...
M5
M0
G1 X10 Y0 F500
G122 W1
M0
G54
G43 H1
S10000 M3
…
```

1. M5 (Spindle Stop). This step stops the spindle so that it can be removed.
2. M0 (Programmed Stop). This step suspends program execution to allow the operator to remove the spindle, install the laser, and close the lid.
3. G1 X10 Y0 F500 (Move to measurement position). This step moves the spindle to the desired measurement position. In this case, the desired position is X10 Y0.
4. G122 W1 (Workpiece measurement cycle). This step executes the workpiece measurement and saves the measurements to the desired offset register. In this case, the measurements are saved to G54.
5. M0 (Programmed Stop). This step suspends program execution to allow the operator to remove the laser, install the spindle, and close the lid.
6. G54 (Apply workpiece offset). This step applies the appropriate workpiece offset.

7. G43 H1 (Apply tool length offset). This step applies the appropriate tool length offset. In this case, the offset for Tool 1 is applied. A current tool measurement is required prior to this step to ensure that the proper tool length offset is applied.
8. S10000 M3 (Start spindle rotation). This step restarts the spindle rotation at the specified speed. In this case the spindle speed is set to 10,000 rpm

Below are the steps that the operator must follow when performing a workpiece sensing operation within the NC code:

1. During first programmed stop – Remove spindle, install laser, and close lid
2. Press start to resume program operation (start workpiece measurement cycle)
3. During second programmed stop – Remove laser, install spindle, and close lid
4. Press start to resume program operation.

Spindle Control Unit

60,000 RPM Spindle Control Unit (for use with electric motor driven spindles)

Use the Nakanishi spindle control unit operation manual in conjunction with these guidelines. If the spindle control unit is displaying an error code, refer to the Nakanishi manual or to tables 5.1 and 5.2 to determine the cause for the error. Use the guidelines below to help troubleshoot the problem.

- **E-stop error** – The E-stop circuit may be tripped. Make sure that the E-stop button on the handheld pendant is not pushed.

- **Air supply error** – The cooling air is not adequate for the spindle. Check the spindle cooling air pressure gauge on the side of the machine as shown in Figure 5.8 to make sure pressure is at least 71 psi.

- **Difficulty clearing errors** – If you think that all the potential causes for errors have been checked, try restarting the spindle control unit by cycling power to the spindle control unit. To cycle power to the spindle, open the Program Run Tab within the CNC Control Panel. Click the *Disable Spindle* button to power off the spindle control unit. Press the *Enable Spindle* button to power on the spindle control unit.

The Nakanishi spindle control unit is located inside the machine above the pneumatic regulators.

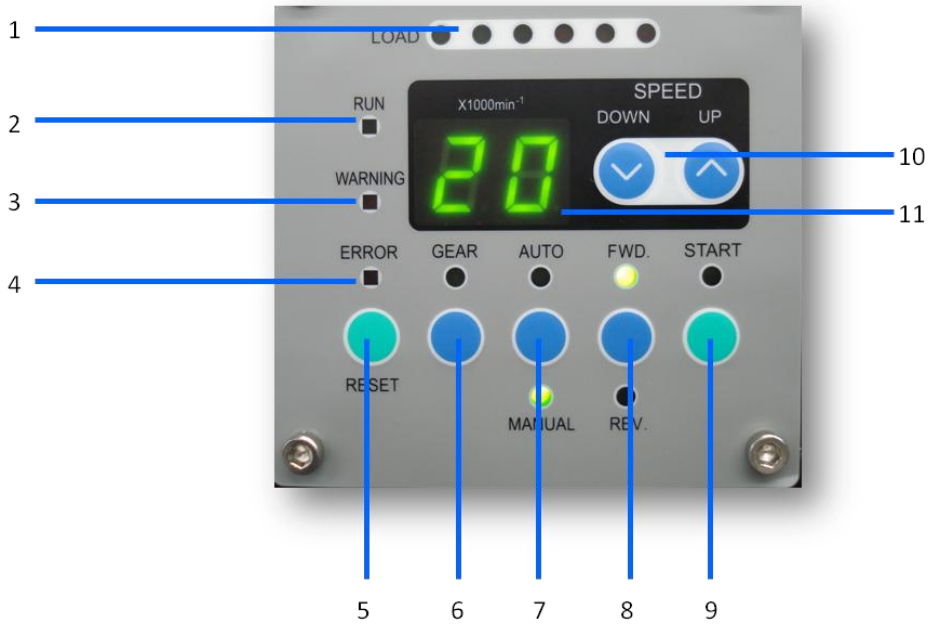


Figure 5.7: 60,000 RPM spindle control unit

The following items reference Figure 5.7.

1. Load Meter
   This meter monitors the load on the spindle. If the yellow or red LEDs are lit, an overload situation exists and the controller will automatically shut down if the situation persists.

   CAUTION: Continued operation under overload conditions will result in overheating and may lead to spindle or controller damage.

2. Run LED
   This light will flash when the spindle is operating.

3. Warning LED
   The Warning LED will flash when a hazardous condition has been detected.

4. Error LED
   The error LED will blink when a serious error is detected. The spindle will automatically shutdown.

5. Error Reset Button
   This button will reset and allow restarting of the spindle after an error has been corrected. Some errors cannot be reset until after the power to the spindle control unit has been turned off. To cycle power to the spindle control unit, open the Program Run Tab within the CNC Control Panel. Click the *Disable Spindle* button to turn off power to the spindle control unit. Press the *Enable Spindle* button to power on the spindle unit.

6. Gear Mode Select Button
   Set Gear Mode to display the rotating speed of the cutting tool when using an angle spindle (not applicable to the 363-S) or speed reducer. Select the gear ratio by using the Speed Adjustment Buttons (10). When setting gear mode, the gear ratio is displayed in the Digital Speed Indicator (11).

7. Controller Button
   This button toggles spindle control between manual mode [using the speed adjustment buttons (10)] and automatic mode (using the CNC Control Panel).

8. Rotation Direction Button
   This button toggles between clockwise (FWD.) and counter-clockwise (REV.) cutting tool rotation as seen when the tool is facing away from the operator.

9. Start Button
   Starts and stops motor rotation.

10. Speed Adjustment Buttons
    These buttons increase (UP) or decrease (DOWN) the motor speed between 5,000-60,000 min^{-1} (rpm).

11. Digital Speed Indicator
    Preset speed, actual speed, warning and error codes are displayed to 2 digits. When the motor is stopped the preset speed is displayed. When the motor is rotating, the actual speed is displayed. The display also displays error and warning codes, as well as the gear ratio while in gear mode.

Table 5.1: Spindle Warning Codes

Warning Code	Warning Function	Trouble
A0	Motor Code	Motor cord or connector disconnected or misaligned
A1	Low Air Pressure	Low air pressure
A2	Overheat Inside Control Unit	Internal temperature of the control unit too high
A3	Over Load	Motor torque load exceeding safe limits
A4	Emergency Stop	Emergency stop system activated
A5	Over Air Pressure	Over air supply

Table 5.2: Spindle Error Codes

Error Code	Problem Area	Trouble
E1	Excess Current	Motor current beyond safe limits
E2	Over Voltage	Motor voltage beyond safe limits
E3	Motor Sensor	Trouble with the sensor signal in the motor
E4	Control Unit Overheat	Internal temperature of the control unit too high
E5	Brake Circuit Trouble	Trouble with the motor brake circuit
E6	Rotor Lock	Motor stalled for more than 3 seconds
E7	Low Air Pressure	Inadequate air supplied for more than 4 seconds during rotation or inadequate air supply when motor start commanded
E8	Torque Overload	Torque limits exceeded for too long a period of time
E9	Communication Interception	Intercept communication with selector unit (NE212) (Only if using NE211 to connect to the system
EA	External Control Signal Error	External control sequencing problem
EC	Internal Memory Error	Trouble with memory (EEPROM)
EE	Emergency Stop Error	Safety relay has been activated and the emergency stop system has stopped the motor
EH	Over Speed	Rotating speed is beyond set speed for too long
EL	Incompatible Motor	Incompatible motor is connected to the system

Part Pattern Tab

The Part Pattern Tab (see Figure 4.11) allows a G-code program to be patterned into an array. The tab enables the user to set pattern size and visualize the array before starting the machining process. Additionally, the program allows the user to set the start and end indices of the array so that certain sections of the array can be machined without machining the whole array. (This feature is useful if the user wishes to machine the first part in the array and inspect it before machining the entire array.) Table 4.3 lists the buttons located in the Part Pattern Tab as well as the function of each button. For a step –by – step list of instructions on using the Part Pattern Tab, refer to the Operation section of this manual.

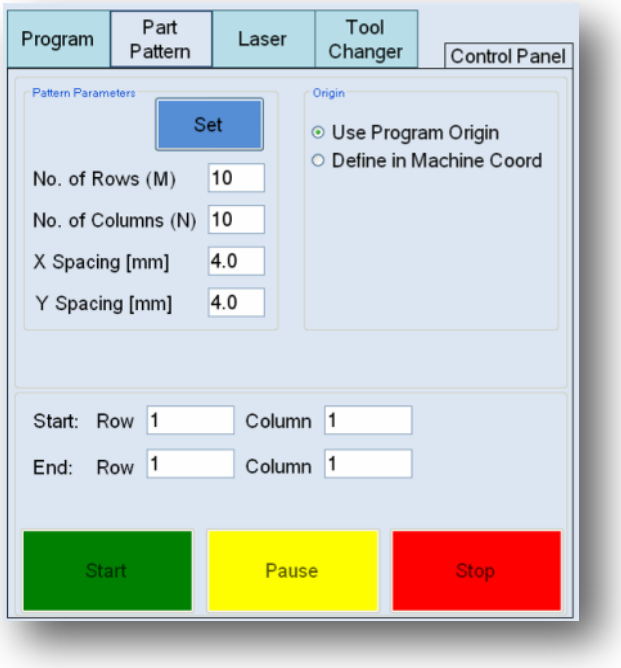


Figure 4.11: Part Pattern Tab

Note: Patterning works by automatically changing the program's X and Y axes zero position by the offset values specified and rerunning the loaded G-code program. The graphical representation of the pattern draws circles with their centers at the origin of that instance of the part.

Caution: The graphical representation does not check the G-code program in any way. The user must verify that the X and Y spacing is large enough so that the individual parts within the array do not overlap. The Z-axis origin is not changed during patterning.

Note: The patterning sequence proceeds from left to right and then top to bottom.

Table 4.3: Part Pattern Tab buttons and their functions

Button	Function
No. of Rows (M)	Specifies the number of rows of parts in the pattern.
No. of Columns (N)	Specifies the number of columns of parts in the pattern.
X Spacing (x)	The desired distance between the centers of adjacent parts in the x direction.
Y Spacing (y)	The desired distance between the centers of adjacent parts in the y direction.
Origin	Enter the location of the top left corner of the pattern. Use current program zero or define new coordinates.
Set	Applies the five pattern parameters above to the program and visualization. The visualization of the array will be displayed in the Axis Tab.
Start Index (Is, Js)	Allows pattern to be started from locations other than the Top Left part. This is useful when a program needs to be stopped after making several parts, to change a tool for example, without throwing away the remaining material or altering the pattern.
End Index (Ie, Je)	Specifies which part to stop the program on, patterns always proceed from left to right and then top to bottom. For example a start index of (1,1) and end index of (2,1) will make one entire row of parts plus the (2,1) part, however a start index of (1,1) and end index of (1,2) will make only two parts.
Start	Starts the part patterning program.
Stop	Stops the part patterning program.
Pause	Pauses the pattern program.

Creating a Part Pattern

Figure 7.27: Part Pattern Tab within the Control Panel

The Part Pattern Tab (Figure 7.27) allows the user to execute programs in the same way as the Program Run Tab but with the added feature of running the program multiple times with different program origins to create an array of parts.

To load NC programs into the program, click the *Browse* button to open a dialog box and find the G-code file. The file name will appear in the box above the *Browse* button. By clicking the *Add* button, that file name will appear in the box below the A*dd* button; multiple files can be added sequentially to this box. After adding the desired files click the *Combine* button which will take the text of the listed files and append them to each other in the order they are listed, the contents of the combined program are then shown in the G-code Tab of the View Panel. Scroll through the code to verify that the correct sequence is loaded. This code will be executed once for each part in the array.

The array parameters can be set using the Pattern Parameters in the upper left corner of the Part Pattern Tab. Hitting *Set* stores the values and updates the display below. The Start Index and Stop Index are adjusted at the bottom right. Details on the functionality of each button within the Part Pattern Tab can be found in the Part Pattern Tab segment of this manual.

Clicking the *Start* button will start the program execution at the part listed in the Start Index. After the program finishes execution on the End Index part the program will stop. Pressing the *Stop* button will also halt program execution.

Example:
The pattern in Figure 7.28 is desired on the surface of a part:

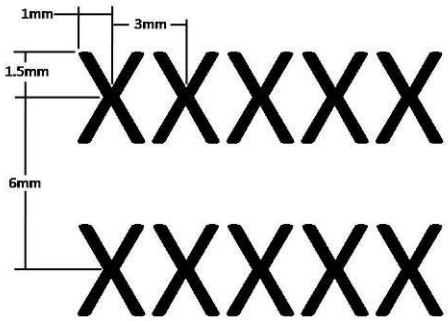


Figure 7.28: Example Part Pattern

Assume the NC program makes each X in two parts. First, the program creates the first diagonal (\ ) and then it makes the second diagonal (/). Each X is 3mm tall and 2mm wide, the spacing between the features is 1mm in the horizontal direction and 3mm in the vertical direction. Each feature is 0.25mm deep and will be machined with a flat end mill 0.25mm wide which is the width of the features. The upper left corner of this pattern should be at X=-5, Y=10 in machine coordinates. For this program the origin is the center of the X at the surface of the material to be removed.

Note: Feedrates are ignored for simplicity in this example.

The first NC program looks like this:

```
N1  G0  X-1 Y1.5      ;Move tool to starting location.
N2  G1 Z-0.25         ;Plunge tool into work piece.
N3  G1 X1 Y-1.5       ;Cut ( \ ).
N4  G0 Z1             ;Pull tool from work piece.
```

The second NC program looks like this:

```
N1  G0 X1 Y1.5        ;Move tool to starting location.
N2  G1 Z -0.25        ;Plunge tool into work piece.
N3  G1 X-1 Y-1.5      ;Cut ( / ).
N4  G0 Z1             ;Pull tool from work piece.
```

The following is a step by step set of instructions that lead the user through the setup process of creating an array within the Part Pattern Tab of the CNC Control Panel.

1. Set up the pattern parameters using the Part Pattern Tab of the CNC Control Panel.

Number of row:	2	
Number of columns:	5	
X spacing (center to center)(mm): 3	[2mm width + 1mm between each column]	
Y spacing (center to center)(mm) 6	[3mm height + 3mm between each row]	
Origin X (mm)	-4	[-5mm + 1mm from center to left edge]
Origin Y (mm)	8.5	[10mm – 1.5mm from center to top edge]

Example Continued:

2. Click the *Set* button and the chart in the Axis Tab of the View Panel will show ovals depicting the location and spacing of the array (Figure 7.29).

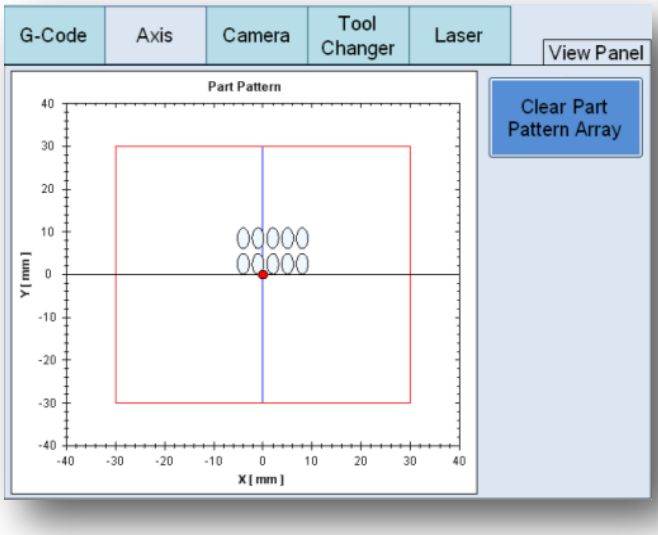


Figure 7.29: Part Pattern Tab

3. Use the *Browse* button to find the NC program for the (\ ). Click the *Add* button to add it to the sequence. Use the *Browse* button to find the NC program for the (/). Click the *Add* button to add it to the sequence. Click the *Combine* button to create the combined NC program, which will be displayed in the G-code Tab of the View Panel.

4. Set the start and end index

   Start Index: (1,1)
   End Index: (2,5)

5. Change to the Program Run Tab to enable the spindle, set its speed, and start the spindle. Change back to the Part Pattern Tab and click the *Start* button to start the machining process.

Legacy Cleaning / Maintenance

Filter Bag Inspection

The coolant filter is located at the rear of the machine. The coolant filter bag should be inspected once a month under typical use. Under heavy use or in processes that create a large quantity of chips, the filter should be inspected more frequently. To inspect the filter bag remove the filter housing lid by unfastening the C-clamp. Remove the lid and inspect the filter bag. If the filter bag is full, it needs to be replaced.

Filter Bag Replacement

13. Remove the lid of the filter housing.
14. Drain the coolant by placing a pan or bucket under the coolant housing and unscrewing the drain stopper at the bottom of the housing.
15. After all the coolant has been drained, remove the filter bag by pulling on its handle.
16. Discard the filter bag in the trash.
17. Rinse filter housing with water to ensure no chips remain in the housing.
18. Replace drain stopper after applying a thread tape to eliminate leaking.
19. Discard the old coolant and rinse-water in a safe location.
20. Insert a new Microlution, Inc. supplied filter bag into the housing and make sure the bag makes full contact with the side of the housing.
21. Add new coolant to the filter housing until the coolant level is about three quarters of an inch below the inlet hole.
22. Replace the housing lid.

Coolant Change

The coolant should be replaced at least as often as the filter bag. If the coolant begins to get foamy before the filter bag needs replacing, the coolant can be changed independently. To change the coolant, follow these instructions:

7. Place a pan or bucket under the filter housing.
8. Unscrew the drain stopper at the bottom of the housing.
9. When all the coolant has drained, add thread locking tape to the drain stopper and screw it back on.
10. Remove the housing lid and add new coolant until the coolant level is approximately three quarters of an inch below the inlet hole.
11. Replace the housing lid.

Protective Rubber Boot Inspection

Frequently check the protective rubber boots for rips and cracks. If a rip or crack is found, immediately discontinue use and contact Microlution, Inc. for a replacement set.

CAUTION: Operating the machine with a ripped or cracked protective rubber boot can allow chips and coolant to enter the machine and cause severe damage to the motion control and electronic systems.

Laser Inspection (LT-9010 Confocal Laser Option Required)

Use the Laser Inspection Tab to measure machined parts without removing the pallet from the machine. After completing the laser inspection process, use the Metrology software to analyze the surface roughness, take measurement, and assess the quality of machined parts. The Laser Inspection Tab is displayed in Figure 7.32.

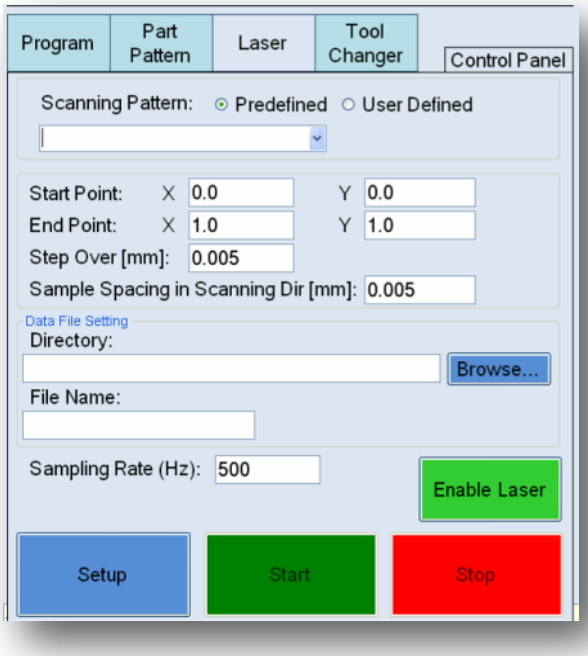


Figure 7.32: Laser Inspection Tab

Laser scans can be performed in one of four predefined scanning patterns or in a user created scanning pattern. The predefined scanning patterns will be the easiest and most useful scanning solutions for typical users.

Scanning Using a Predefined Scanning Pattern

1. Mount the laser distance sensor to the Z stage (remove the spindle if necessary).
2. Open the Laser Inspection Tab within the CNC Control Panel.
3. Select the Predefined Scanning Pattern selection circle, Figure 7.32. From the drop down menu, select the desired scanning pattern. Table 7.3 describes the scanning patterns available in the drop down menu.

Table 7.3: Predefined Scanning Patterns

Scan Type	Description
Rectangular Area (X Scan)	Sets the scan to proceed from the start X and Y position to the end X position. After reaching the end X position, the Y position is incremented towards the Y end position by the step amount and scans from the end X value to the start X value. This process repeats until the end value of X and Y is reached.
Rectangular Area (Y Scan)	Sets the scan to proceed from the start X and Y position to the end Y position. After reaching the end Y position the X position is incremented towards the X end position by the step amount and scans from the end Y value to the start Y value. This process repeats until the end value of X and Y is reached.
Circular Area (R Scan)	The scan will move radially starting at the start radius and start angle until the end radius is reached. The angle will be incremented by the step amount and the scan will proceed along the new radius until reaching the start radius. This process repeats until the end value of radius and angle is reached.
Circular Area (Arc Scan)	The scan will move along an arc starting with a radius equal to the start radius and angle equal to the start angle. The angle will increment until the end angle is reached. The radius will then be incremented by step amount. The scan will then decrement the angle until the start angle is reached. This process repeats until the end value of radius and angle is reached.

4. Choose the X and Y program coordinates where the scan will start and stop, refer to Figure 7.32 for button locations. For circular scan patterns, choose the X and Y program coordinates of the center of the scanning arc (circle), and the radii of the start and stop arcs (circles). The scan path will progress based on the selected scanning pattern.
5. Choose the Step Over distance. The Step Over distance is the distance the laser moves after completing one line of scanning. It is recommended to set the Step Over distance equal to the Sample Spacing.
6. Choose the desired Sample Spacing. The Sample Spacing is equivalent to the distance between each data point in the scanning direction.
7. Choose the Sample Rate. The Sampling Rate decides how many data points are taken each second. The computer computes the necessary feedrate based on the Sample Spacing and the Sampling Rate.
8. Use the *Browse* button to choose the directory the file is saved to.
9. Name the file using the File Name box.
10. Press the *Enable Laser* button to turn the laser on.
11. Press the *Setup* button to prepare the laser.
12. Press the *Start* button to start scanning.
13. A dialog box will appear to inform the user when the scan is finished.
14. The *Stop* button can be pressed at any time to stop the scanning process if necessary.

Scanning Using a User Defined Scanning Pattern

1. Select the User Defined Scanning Pattern selection circle, displayed in Figure 7.33.

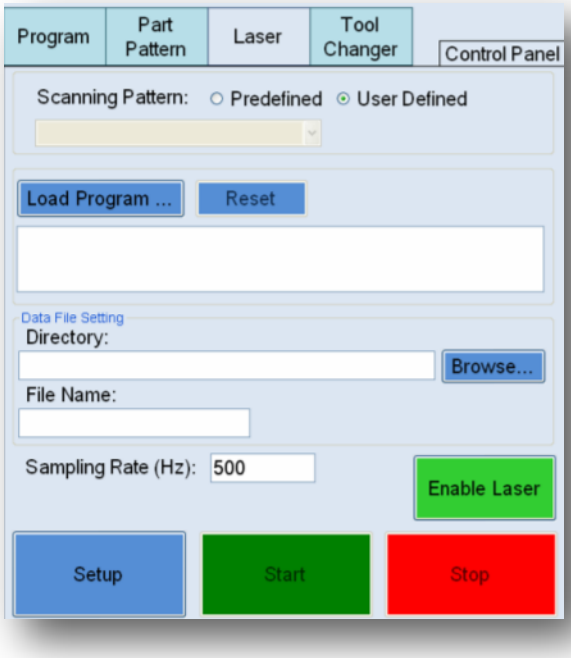


Figure 7.33: User Defined Scanning Program within the Laser Inspection Tab

2. Press the *Load Program* button to open a user-created G-code scanning pattern.
3. Choose the Sampling Rate. The computer determines the sample spacing based on the G-code-specified feedrate and the user-specified sampling rate.
4. Press the *Browse* button and select which directory to save the scan to.
5. Name the file using the File Name text box.
6. Press the *Setup* button to prepare the laser and load the G-code.
7. Press the *Start* button to start scanning.
8. A dialog box will appear to inform the user when the scan is finished.
9. The *Stop* button can be pressed at any time to stop the scanning process if necessary.

User I/O (option required)

The User I/O option provides 6 digital Inputs, 8 digital outputs, 2 Analog Inputs, 2 Filtered PWM Analog Outputs, and access to 4 User Flags. Table 7.4 and Table 7.5 show the pinout of the two user I/O breakout boards. A description of each type of input is described below.

Table 7.4: Connector 1 (DB-25 breakout) pinout

Pin	Description	Software Access Variable
1	NC	
2	Digital Input 2	M26
3	NC	
4	Digital Input 4	M30
5	Digital Input 6	M28
6	Digital Output 2	P2022
7	Digital Output 4	P2024
8	Digital Output 6	P2026
9	Digital Ouptut 8	P2028
10	Analog Input 1-	M5064
11	Analog Input 2-	M5063
12	Analog Output 2+	M802
13	NC	
14	Digital Input 1	M27
15	Digital Input 3	M25
16	NC	
17	Digital Input 5	M29
18	Digital Output 1	P2021
19	Digital Output 3	P2023
20	Digital Output 5	P2025
21	Digital Output 7	P2027
22	Analog Input 1+	M5064
23	Analog Input 2+	M5063
24	Analog Output 1+	M702
25	COM (0V)	

Table 7.5: Connector 2 (DB-9 breakout) pinout

Pin	Description	Software Access Variable
1	User Flag 1+	M117
2	User Flag 2+	M217
3	User Flag 3+	M317
4	User Flag 4+	M417
5	NC	
6	User Flag 1-	M117
7	User Flag 2-	M217
8	User Flag 3-	M317
9	User Flag 4-	M417

Digital Inputs

The digital inputs accept 24V single-ended input signals. The digital inputs can be accessed in the controller by accessing the appropriate M-variable (see Delta Tau User Manuals: GeoBrick, Turbo PMAC User Manual, and Software Reference for more information) shown in the table above.

Digital Outputs

The digital outputs are open collector (sinking) outputs. They require an external positive voltage supply that has its common tied to the machine common (Pin 25 of Connector 1). Each output has a maximum sink current of 100mA. The digital outputs are controlled by a multiplexing circuit within the Microlution 363-S. The states of the outputs are updated with each execution of a multiplexing PLC (approx. 100ms). A partial schematic of the output circuit is shown below.

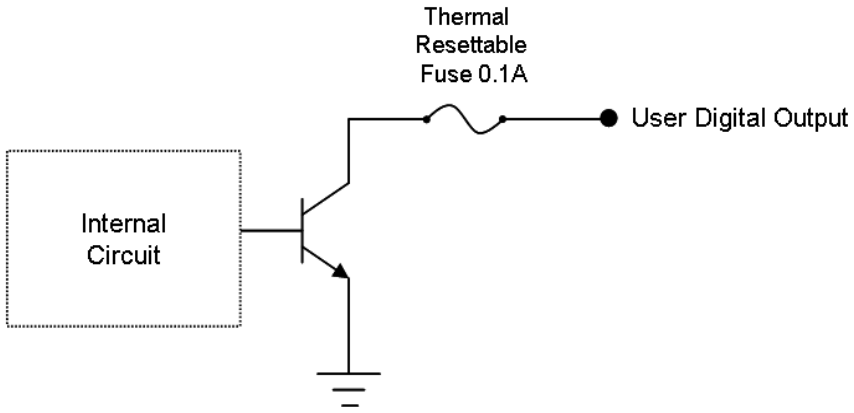


Figure 34: Digital output partial schematic

Analog Inputs

Analog inputs are wired directly to Analog Inputs 3 and 4 on the GeoBrick controller. Each input accepts +/-5V single-ended or differential inputs. The voltage can be read using the proper software variables (see Table 1) using the scaling factor below. See the Delta Tau GeoBrick User Manual for more details.

Scaling:
V_{in} = M506x * 5V / 32,768

Example - M5063 = 14500 (User Analog Input 1)
V_{in} = 14,500 * 5V / 32,768 = 2.213V

Filtered PWM Analog Outputs

The analog outputs are wired directly to analog outputs 3 and 4 on the GeoBrick controller. The outputs are +/- 10V single-ended. The output voltage can be set by setting the proper software variable (see Table 1) using the scaling factor below. See Delta Tau GeoBrick User Manual for more details about the Analog Outputs.

Scaling:
V_{out} = 10V * (Mx02) / 810

Example – Set Analog Output 1 to -5.5V:
M702 = -5.5V / 10V * 810 = 445.5

User Flags

The user flags are wired through an interface circuit to the user input flags for each motor on the GeoBrick controller. The flags accept 24VDC logic. The flag input voltage can be wired with either positive or negative polarity with respect the descriptions in Table 2. Note: the User Flag 1-, User Flag 2-, User Flag 3-, and User Flag 4- inputs are not common. Please refer to the Delta Tau GeoBrick User Manual and Delta Tau Software Reference Manual for more information on how to use flags.